ADAMA SCIENCE AND TECHNOLOGY UNIVERSITY
SCHOOL OF ENGINEERING AND INFORMATION TECHNOLOGY

DEVELOPMENT OF STRUCTURAL ANALYSIS AND DESIGN SOFTWARE TO EBCS – 1995

(Beam, Column and Footing)

IN

PARTIAL FULFILLMENT OF B.Sc DEGREE IN **CIVIL ENGINEERING**

SPECIALIZED IN **STRUCTURAL ENGINEERING**

BY:

ABIY FANTAYE MELAKU

TSINUEL NURILLIGNE GELETA

ZEINEBA MEHEDI SULEMAN

ADVISOR:

AYELE ZEWDU

____

EXAMINERS

____

____

June, 2013

Adama, Ethiopia

EESADS V 1.0

Ethiopian Structural Analysis and Design Software

Dedicated To

AYELE TESFAYE ADMASU

Our Physics teacher back in high school

And who encouraged us to start reading on Software Engineering

Acknowledgement

We hereby would like to express our gratitude to **Mr. Ayele Zewdu**, our advisor, for his willingness to advise us through the whole project. He was helping us in figuring the issues we need to focus and to meet the requirements of senior project.

We are also thankful for **Mr Mustofa Aman**, former Civil Engineering dep't head, for his heartfelt willingness to explain the importance of our project to the higher university officials.

In general, we want to show our appreciation to our friends, parents, teachers and everyone who was giving us constructive criticism. We wouldn't have achieved this much if it had not been for their contribution.

The Project Team

How to Use the Document

This document is prepared as a Senior Project Report for Software Development. In order to ease up the navigation through the document and meet the ultimate goal of the document, this section gives some simple guidelines of using the document.

Parts and Chapters

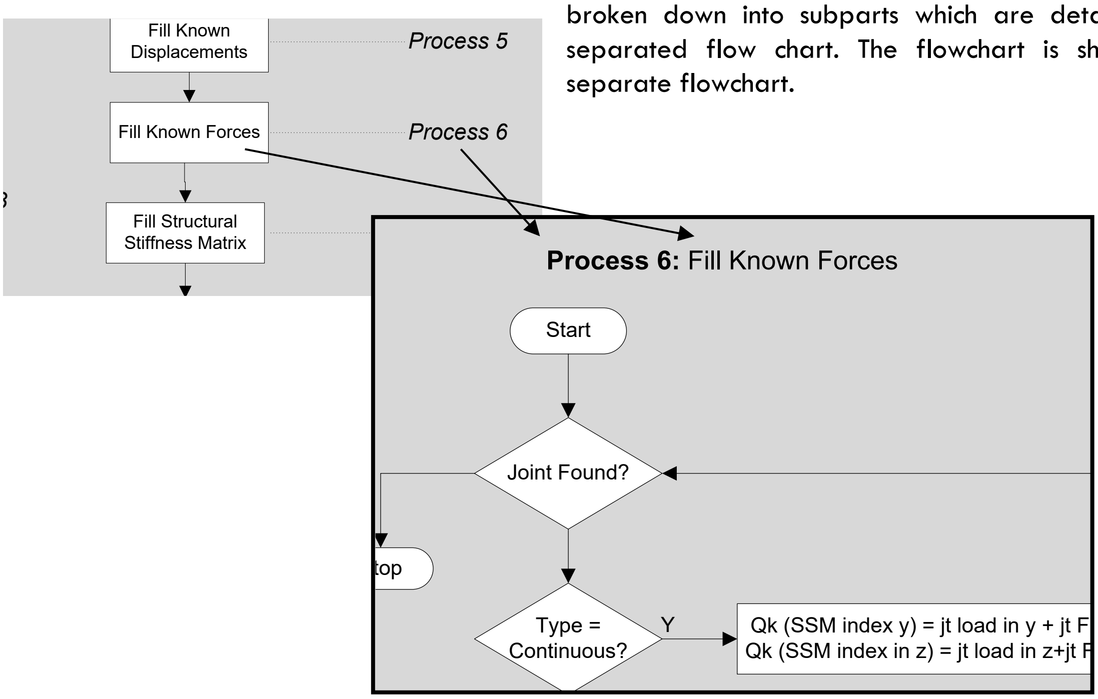
The document has a total of four main parts: ESADS in General, Continuous Beam, Column and Footing. The first part is concerned with stating the general issues which are not specific to any other component. Those things given in this part apply to all other parts as well.

The other three parts are specific to the component they are concerned with. These parts only discuss their specializations. Everything they have in common with other components is discussed in part I only. There are some parts of the document outside the four parts. The Recommendation and Appendix chapters are exemplary.

Each chapter and part starts with an overview of itself that may serves the user as a map to navigate through the chapter or part.

Visuals

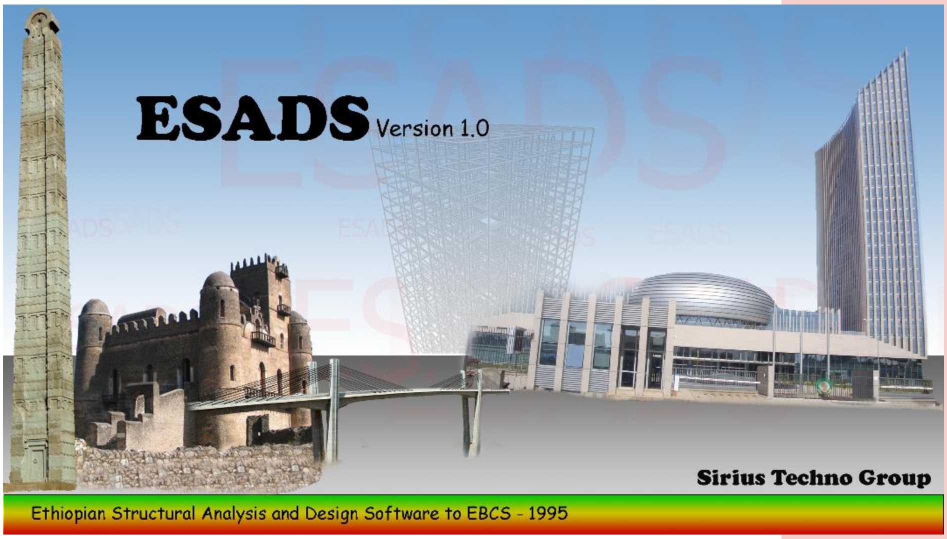
Flowcharts, activity diagrams and schematic diagrams are frequent throughout the document. Flowcharts are scientific diagrams used to describe a scientific process. Since most procedures involve long process, the flowcharts are broken down into subparts which are detailed in separated flow chart. The flowchart is shown in separate flowchart.



When it is important to use activity diagram, it is presented next to the corresponding use case. Most of the time activity diagrams are presented for complicated processes.

Contents in Brief

Table of Contents



The Startup Dialog of ESADS v1.0
Courtesy: The developers' team
In the picture, the designer wants to say ''ESADS links (the bridge) old Ethiopian (Fasil and Axum) to the modern construction (The new AU head quarter).

PART I ESADS IN GENERAL

Chapters:

- Introduction
- Building Codes and Standards
- Software Engineering – An Overview

ESADS V 1.0

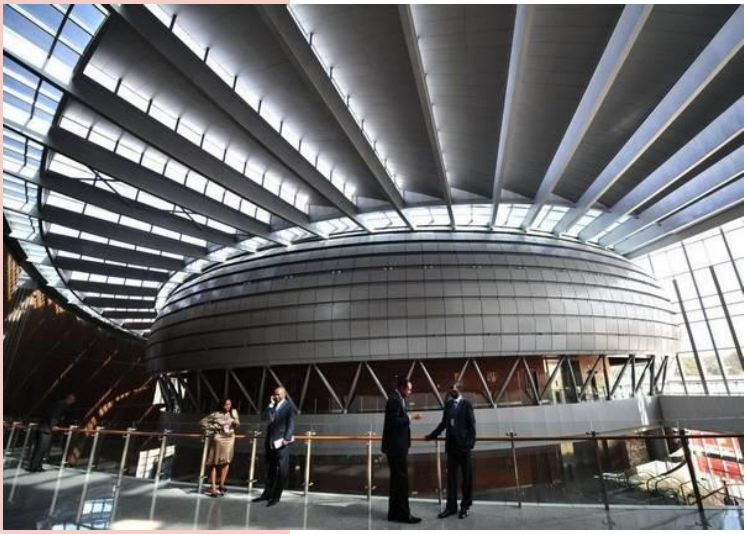
This part of the document gives the general concerns about the project and software development totally. In the introduction, most frequently used terms are defined, the statement of the problem is then stated followed by the goals and objectives, scope and finally the challenges faced in working with the project.

In the next chapter, a brief description of structural analysis and design is given. The chapter starts by defining what structure means. After that structural analysis and design concepts are discussed including their inter-relation. The chapter ends by giving the basic and general provisions of EBCS applied in the development of the software.

The third chapter of this part is an over-view of Software Engineering and the software development process. It gives the basics of the development process of software.

Finally, this part concludes by giving the development approach of ESADS and presenting the general features of it applicable to all of its components. Use cases of the key features are included in this chapter.

1. Introduction



Outline:

- Definition of Terms
- What is Structural Analysis and Design?
- Software Usage for Structural Analysis and Design
- Schemes of Structural Analysis and Design
- Need for Solution
- ESADS – As a solution
- Goals and Objectives
- Scope and Features of ESADS v 1.0
- Challenges of the Project

In this chapter we are going to discuss general ideas related to the importance of structural analysis and design software. First definition of some important terms is given. Then, an overview of structural analysis and design is viewed. Afterwards, the current trend of structural analysis and design in Ethiopia is discussed in relation to softwares. Then, the goals and objectives of the project are stated shortly. Most importantly, the scope and features of the software are stated in this chapter which states the overall capabilities of the software are stated briefly here. Finally, the challenges encountered are stated to help future developers, specially, students to know what problems they may face while trying to achieve their goal.

1.1. Definition of Terms

The following are some of the most important terms used frequently throughout this document.

Structure: Organized combination of connected parts designed to provide some measure of rigidity. ISO 6707: Part 1 gives the same definition but it adds "or a construction works having such an arrangement". [EBCS – 1, 1995, Section 1.1.3]

Structural Analysis: This term is used in this document in two different contexts.

Structural Design: is the process of determining the necessary material, size, shape and arrangement of structures based on the maximum load encountered to enable them to stand without failure.

Beam: A structural line element which supports load primarily by bending, shear and torsion. Usually, it is constructed in horizontal or inclined arrangement.

Column: A structural line element which primarily supports axial load. It also supports bending moment, shear and torsion if it is exposed to it.

Footing: A sub-structure used to transfer load from the superstructure to the ground. It is a kind of shallow foundation mostly receiving its load from column.

Slab: A horizontal diaphragm type structural element used to support area loads acting into its plane.

1.2. What is Structural Analysis and Design?

Structural analysis and structural design are two different yet inter-related terms in structural engineering. Structural analysis is the process of determination of internal forces of the structure caused by external loads on the structure. These forces may be shear force, bending moment or torsional moment on the structural member. When we say internal forces, we mean the forces acted on an element taken by cutting the member somewhere.

There are different methods of structural analysis. Some of the commonly practiced methods are Kani's method, moment distribution method, slope deflection method, flexibility method, stiffness method and finite element method. The method adopted depends on the determinacy of the structure if manual method is going to be used. Most methods adopted for computer, however, use the same algorithm for both determinate and indeterminate structures.

The primary use of getting the internal forces of structures is to decide the size strength of the material at that specific location so that it will not fail under the applied load. The process of determining the appropriate material, shape and size of structural member is called Structural Design. Therefore, the primary aim of structural analysis is to 'tell' how much 'danger' is going to come so that the structure gets ready to withstand the 'danger'.

1.3. Software Usage for Structural Analysis and Design

These days almost every activity of human kind is being supported by computers. The versatility of computers has become apparent through the development of application softwares specialized for the final user's need. Professionals throughout the world use different kind of software to facilitate their daily activity. Structural Engineers also use different softwares to aid in the structural analysis and design process.

The process of structural analysis and design should be as per the provision of the local standards. The standards are prepared based on the specific situations of that country. Compliance with this standards is a legal requirement of the designer enforced by some authority of the government.

In structural engineering, there are different variety of softwares used to conduct structural analysis and design of a structure with the designer's 'supervision'. The job shared by the software is increasing with the development in quality of softwares. Most design softwares use graphical input system, instead of numeric input in spreadsheet format.

1.4. Schemes of Structural Analysis and Design in Ethiopia

As a developing country, Ethiopia has a fast growing construction industry. The designers in this construction sector use different methods to ease up their job. One of the best ways to do this is through the use of softwares.

Currently, there is no software having the Ethiopian Building Code of Standard in their code list. This makes it difficult to use it alone due to limitation by law.

Most design offices in Addis Ababa use spreadsheet applications in combination with some foreign softwares. They use the softwares for the analysis of the structures, which is not as much dependent on the building code. The design is done by using spreadsheet applications.

Some of the most commonly applied softwares in Ethiopia are:

- ETABS® by Computers and Structures Inc. (CSI)
- SAP2000® by CSI
- SAFE® by CSI
- TEKLA® Structures
- Revit® Structures by Autodesk®
- Robot® Structures by Autodesk®
- Bentley® STAAD® Pro
- Design Expert

1.5. Need for Solution

Analysis and design of buildings in Ethiopia is so variant that the custom method of design is adopted in most design offices. This is mainly because they don't get a consistent software that does the whole work from the start to the end. Designers need software that works with EBCS. All the softwares mentioned above are not able to design using The Ethiopian Building Code of Standard.

Therefore, the problem statement may is:

"There is no fully equipped structural analysis and design software that can be used to conduct building designs as per The Ethiopian Building Code of Standards."

1.6. ESADS – As a Solution

Ethiopian Structural Analysis and Design Software (ESADS) is a software made as senior project in Adama Science and Technology. ESADS is made totally based on The Ethiopian Building Code of Standards (EBCS). This enables the designer to conduct the whole analysis and design process being aided by computer.

ESADS may not be applicable in real design, in its current stage. However, with some testing and improvements, it really will be the best tool to conduct building designs.

Designers had the major problem of getting a software capable of conducting a design software that is able to design using EBCS. ESADS will solve this by taking the part where the code is required so that the gap in softwares will be filled.

Most of all, its compatibility with the existing softwares involved in the construction industry will make it fluent in fulfilling the needs of designers. The designer needs to do the following to finish the design of common buildings. Modelling a building in SAP2000 or ETABS, run the analysis only, export database to ESADS, run the design on ESADS and export the detailing to AutoCAD. This will be true soon, blowing away the nightmare of most Ethiopian designers and finishing the whole work of building design within few hours spent to model it.

If the designer wishes to modify the design or there is something that the software is not capable of doing, he will be able to interfere in the design output and interact so as to modify the output accordingly.

By the time ESADS achieves its long-term goal, even the modelling will be done on ESADS itself. This will enable the user to have a very flexible input mechanisms, modification options and flexible output. By this, it will be internationally competent.

1.7. Goals and Objectives

The basic goal of ESADS relates to the creation of the first idea. The source of the idea was the observation of the problem.

Long Term Goal: *"To develop internationally competing structural analysis and design software that includes EBCS in its code list."*

Short Term Goal: *"To develop structural analysis and design software that works with EBCS for beam, column, footing and slab."*

The objectives of the project as a senior project is stated below.

Objectives:

- To develop an easy to use, efficient and accurate software to design and detail beam, column, footing and slab.
- To excel the in problem solving capacity and ability to observe community problems.
- To initiate students and scholars to develop better methods of analysis and design so as to achieve the goal of designing.

1.8. Scope and Features of ESADS v 1.0

The section states the capabilities of the current version of ESADS and the general overview of its main features. The scope of this version of ESADS may be seen from three different angles. First the size of the structures it is capable of designing is stated. Then the behavior and material of the structures designed by ESADS is presented. Finally, the input and output capabilities of ESADS is discussed.

Structures designed by this version of ESADS are:

- The scope of the Ethiopian Building Code of Standards apply with further limitation of the following.

- All structures are made of Reinforced concrete
- The creep and shrinkage effects of concrete are not considered in the design.
- The modulus of elasticity of concrete is considered using the approximate formula [EBCS – 2, 1995, Eq. 2.3]. In case when the user wants to use more accurately calculated value that may be used directly.
- The only design situation dealt with is Persistent and transient, no accidental design situation is considered. [EBCS – 2, 1995, Sec. 3.5.3.]
- Design is considered in Unfavorable design condition only [EBCS – 2, 1995, Sec. 3.6]
- The bond condition of reinforcements is always considered to be good. [EBCS – 2, Sec. 7.1.5]
- Variation of material is not allowed in a single model.
- Beam should be rectangular prismatic with non-significant axial and torsional load.
- The column to be designed should be loaded with axial load and bending moment only. Columns with torsional moment is not considered.
- ESADS designs rectangular footing loaded with axial load and moment only.

1.9. Challenges of the Project

From the very beginning of the project we faced both tangible and psychological challenges. At the ignition of developing the idea of building software, many people thought that it was not as such important to work on such 'small software' while there are so many sophisticated softwares out there. Some commenters thought that it would be better to use the conventional methods of design than to build such a 'non-competent software'. We always said: "Rome was not built in one day."

The idea of professionalism was also another challenge for us. We did all this work by reading programming and then software engineering concepts by ourselves. This fact posed question on many people – 'Why? Shouldn't this be the job of the software engineers?' It is true that the software engineers are the best men to build a fully equipped and an 'as per standard' software. However, they can never be better than the structural in understanding the way structures behave and how to handle their design. Obviously, rather than sharing an idea to other person and fulfill a need, it is better to do it by oneself.

Even if we read programming merely by interest, it was also a bit challenging to achieve what we wanted on time. More than that, the development approaches of software, software architecture and other ideas related to the formal software development had been really tough. We had to read a lot to get tiny idea related to our work. We couldn't get professionals who can comment on our work contextually; we did our best to achieve this last work mostly by our understanding.

Besides our own problems, we faced some external challenges. At the start of the project we proposed to build a software to design beam, column, footing and slab. There were some requirements that we proposed to be fulfilled to achieve the ultimate goal. The most important one was the assignation advisors from Civil Engineering and Software Engineering departments. Apparently, we couldn't get the advisor from the Software Engineering departments which created an extra burden on us.

Furthermore, the time frame put forth by the university was not followed accordingly. This made us to start late and end it early, creating even an extra problem.

Due to the above two reasons, we were obliged to leave the slab component. We started to work on it up to the finish of analysis modeling, but we couldn't work on the design architecture and construction. All the jobs proposed in cycle two of the development mentioned in the proposal could not be met.

2. Building Codes and Standards

Outline:

- ❑ What are Building Codes and Standards?
- ❑ Basic EBCS Provisions on Analysis and Design of Structures
  - Data on Concrete and Steel
  - Limit States
  - Basis of Design
  - Detailing

The discussion of this chapter begins with introduction to building codes and standards. Then the provisions of EBCS are given in context of this version of ESADS. In this section, data on concrete, limit states, basis of design and detailing provision are the main parts.

2.1. What are Building Codes and Standards?

Almost all countries all over the world prepare standards for different activities and things. They require the personnel involved in the corresponding works to conform to the standards in their every activity.

Building Codes and Standards are statements of requirements in the design and construction of buildings. They are issued and enforced by some government body, The Ministry of Works and Urban Development in the case of Ethiopia.

Mr. Haile Assegidie, minister (in 1995) of The Ministry of Works and Urban Development, in the foreword section of every EBCS book, says:

> *"…The purpose of these standards is to serve as nationally recognized documents, the application of which is deemed to ensure compliance of buildings with the minimum requirements for design, construction and quality of materials set down by … The major benefits to be gained in applying these standards are the harmonization of professional practice and the ensuring of appropriate levels of safety, health and economy with due consideration of the objective conditions and needs of the country."*

Which means the primary purpose of building codes is for the safety and uniformity of design activities. Therefore, it is an obligation to conform to this code while conducting any design activity.Basic EBCS Provisions on Analysis and Design of Structures

This section presents the requirements of EBCS that are important for the work of ESADS. The following articles are directly taken from the specified sections of EBCS.

2.1.1. Data on Concrete and Steel [EBCS – 2, 1995, Chapter 2]

Concrete

Concrete is graded in terms of its characteristic compressive cube strength. The grade of concrete to be used in design depends on the classification of the concrete work and the intended use.

The following table gives the permissible grades of concrete for the two classes of concrete works.

The numbers in the grade designation denote the specified characteristic compressive strength in MPa.

Class	Permissible grades of Concrete
I	C5, C15, C20, C25, C30, C40, C50, C60
II	C5, C15, C20

Grade C5 shall only be used as lean concrete.

The characteristic cylindrical compressive strength f_{ck} are given for different grades of concrete. Table 2.3 of EBCS – 2 gives these values as:

Grades of Concrete	C15	C20	C25	C30	C40	C50	C60
f_{ch}	12	16	20	24	32	40	48

In the absence of more accurate data, the characteristic tensile strength may also be determined from the characteristic cylinder compressive strength according to Eq. 2.1. of EBCS – 2.

$$f_{ctk} = 0.7 f_{ctm} \qquad 2\text{-}1$$

Where f_{ctm} is the mean value given by Eq. 2.2. of EBCS – 2.

$$f_{ctm} = 0.3 f_{ctk}^{\frac{2}{3}} \qquad 2\text{-}2$$

In the absence of more accurate data, or in case where great accuracy is not required, an estimate of the secant modulus, E_{cm} can be obtained by:

$$E_{cm} = 9.5(f_{ck} + 8)^{\frac{1}{3}} \qquad 2\text{-}3$$

Where E_{cm} is in GPa and f_{ck} is in MPa. They relate to concrete cured under normal conditions and with aggregates predominantly consisting of quartzite gravel. When deflections are of great importance, tests shall be carried out on concrete made with the aggregate to be used in the structure. In other cases experience with a particular aggregate, backed by general test data, will often provide a reliable value for E_{cm}, but with unknown aggregates, it would be advisable to consider a range of values.

Steel

The following are some of the important physical characteristics of steel.

- The density of steel may be assumed as a mean value of 7850 kg/m^3.
- The mean value of modulus of elasticity E, may be assumed as 200 GPa.

Design Strength [EBCS – 2, 1995, Sec. 3.5.2]

The design strength for a given material property and limit state is obtained, in principle, by dividing the characteristic strength by the appropriate partial safety factor for the material property, γ_m, i.e.

$$f_d = \frac{f_k}{\gamma_m} \quad \text{2-4}$$

However, in the case of concrete under compression, a further correction factor is introduced in this Code for convenience.

The design strength of concrete is defined by:

In compression:

$$f_{cd} = \frac{0.85 f_{ck}}{\gamma_c} \quad \text{2-5}$$

In tension:

$$f_{ctd} = \frac{f_{ctk}}{\gamma_c} \quad \text{2-6}$$

The design strength of steel in tension and compression is given by:

$$f_{yd} = \frac{f_{yk}}{\gamma_s} \quad \text{2-7}$$

Partial Safety Factors for Materials [EBCS – 2, 1995, Sec. 3.5.3]

Merging Table 3.1 and Table 3.2 of EBCS 2 to eliminate accidental design situation, we get:

Table 2-1: Partial Safety Factors of Materials in Persistent and Transient Design Situation

Class of Work	**Partial safety factor in persistent and transient design situation**	
	Concrete	**Steel**
Class I	1.5	1.15
Class II	1.65	1.2

2.1.2. Limit States [EBCS – 2, 1995, Sec. 3.2]

A structure, or part of a structure, is considered unfit for use when it exceeds a particular state, called a limit state, beyond which it infringes one of the criteria governing its performance or use.

The limit states can be placed in two categories:

a. **The Ultimate Limit States** are those associated with collapse, or with other forms of structural failure which may endanger the safety of people. States prior to structural collapse which, for simplicity, are considered in place of the collapse itself are also treated as ultimate limit states.

The ultimate limit states which may require consideration include:

- Loss of equilibrium of a part or the whole of the structure considered as a rigid body.
- Failure by excessive deformation, rupture or loss of stability of the structure or any part of it, including supports and foundations.

b. **The Serviceability Limit States** correspond to states beyond which specified service requirements are no longer met.

Serviceability limit states which may require consideration include:

- Deformations or deflections which affect the appearance or effective use of the structure (including the malfunction of machines or services) or cause damage to finishes of non-structural elements.
- Vibration which causes discomfort to people, damage to the building or its contents, or which limits its functional effectiveness.
- Cracking of the concrete which is likely to affect appearance, durability or water tightness adversely.

Partial Safety Factors for Actions [EBCS – 2, 1995, Sec. 3.6.1]

For actions in Unfavorable design condition and persistent and transient design situation, the factors are:

Permanent Action: 1.3

Variable Action: 1.6

2.1.3. Basis of Design

The calculation of the ultimate resistance of members for flexure and axial loads shall be based on the following assumptions, in addition to those given in Sections 3.7 and 3.8 of EBCS - 2

- Plane sections remain plane
- The reinforcement is subjected to the same variations in strain as the adjacent concrete
- The tensile strength of the concrete is neglected
- The maximum compressive strain in the concrete is taken to be:

  0.0035 in bending (simple or compound),

  0.002 in axial compression

- The maximum tensile strain in the reinforcement is taken to be 0.01.

Referring to Fig 4.1 of EBCS – 2, the strain diagram shall be assumed to pass through one of the three points A, B or C.

The parabolic-rectangular stress distribution shown in Fig 4.2 of EBCS – 2 may be used for calculation of section capacity.

For sections which are partly in tension (beams or columns with large eccentricity), the simplified stress block shown in Fig 4.3 of EBCS – 2 may be used.

N.B. The following four figures are those taken directly from EBCS. Even the figure numbers are those of EBCS's.

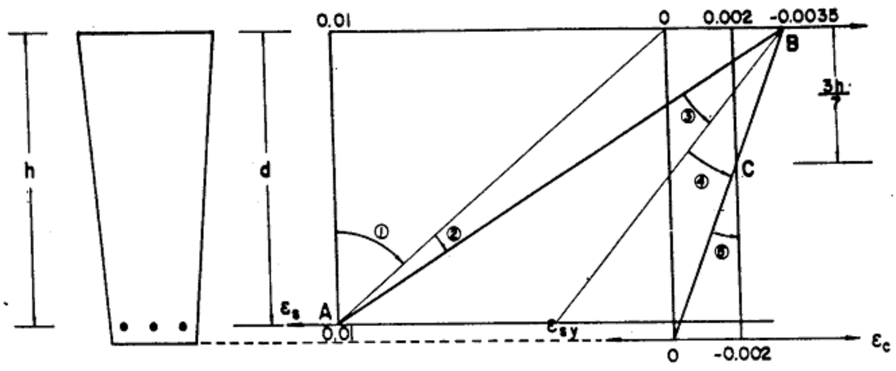


Figure 4.1 Strain Diagram in the Ultimate Limit State

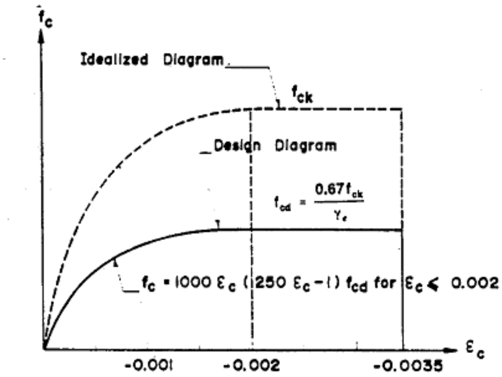


Figure 4.2 Parabolic-Rectanglar Stress-Strain Diagram for Concrete in Compression

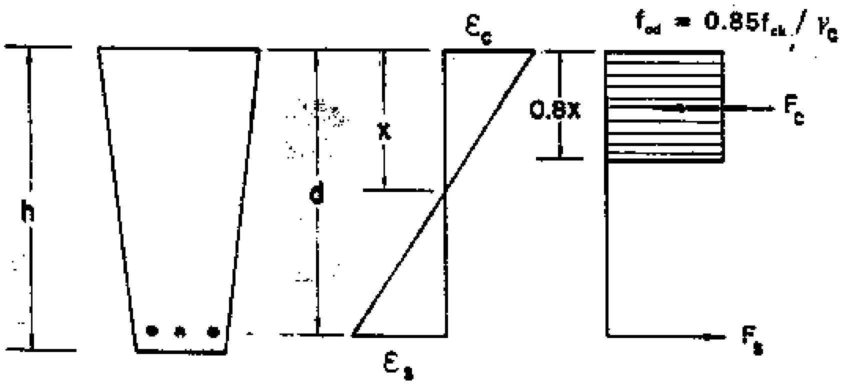


Figure 4.3 Rectangular Stress Diagram

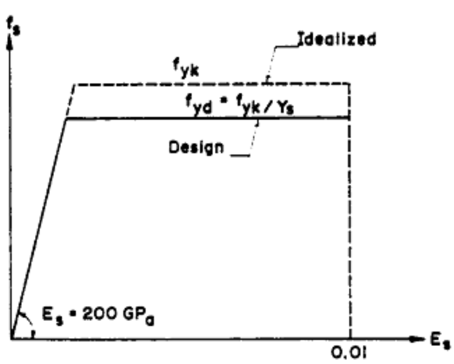


Figure 4.4 Stress-Strain Diagram for Reinforcing Steel

2.1.4. Detailing [EBCS – 2, 1995, Chapter 7]

Concrete Cover to Reinforcement [Section 7.1.3]

The minimum concrete cover to all reinforcement including links and stirrups should not be less than the appropriate values given in Table 7.2 of EBCS.

Table 2-2: Concrete Cover to Reinforcements

Type of exposure	Dry environment: Interior of buildings of normal habitation or offices **(Mild)**	Humid environment: Interior components (e.g. laundries); exterior components; components in non-aggressive soil and/or water **(Moderate)**	Seawater and/or aggressive chemical environment: Components completely or partially sub-merged in seawater; components in saturated salt air; aggressive industrial atmospheres **(Severe)**
Minimum cover (mm)	15	25	50

Spacing of Reinforcement [Section 7.1.4]

The clear horizontal and vertical distance between bars shall be at least equal to the largest of the following values:

a. 20 mm
b. The diameter of the largest bar or effective diameter of the bundle
c. The maximum size of the aggregate d, plus 5 mm

Where bars are positioned in separate horizontal layers, the bars in each layer should be located vertically above each other and the space between the resulting columns of bars should permit the passage of an internal vibrator.

Design Bond Strength [Section 7.1.5.1]

For good bond conditions, the design bond strength of plain bars may be obtained from:

$$f_{bd} = f_{ctd} \tag{2-8}$$

For deformed bars twice the value for plain bars may be used.

For other bond conditions, the design strength may be taken as 0.7 times the value for good bond conditions.

Anchorage of Reinforcement [Section 7.1.6]

The basic anchorage length l_b for a bar of diameter ϕ is:

$$l_b = \frac{\emptyset}{4}\frac{f_{yd}}{f_{bd}} \tag{2-9}$$

The required anchorage length, $l_{b,net}$ is calculated as:

$$l_{b,net} = a l_b \frac{A_{s,calc}}{A_{s,prov}} \geq l_{b,min} \tag{2-10}$$

Where: $A_{s,calc}$ is the theoretical area of reinforcement required by design

$A_{s,ef}$ is the area of reinforcement actually provided a = 1.0 for straight bar anchorage in tension or compression

b = 0.7 for anchorage in tension with standard hooks

$l_{b,min}$ the minimum anchorage length

For bars in tension

$$l_{b,min} = 0.3 l_b \geq 10\emptyset \quad \text{2-11}$$

$$or \ \geq 200mm$$

For bars in compression

$$l_{b,min} = 0.6 l_b \geq 10\emptyset \quad \text{2-12}$$

$$or \ \geq 200mm$$

3. Software Engineering – An Overview

Outline:

- Software Development Process
- Object Oriented Paradigm

In this chapter, we are going to discuss the software engineering in general. The software development process is the first idea discussed in this chapter. Then the chapter concludes by showing what object oriented paradigm means.

3.1. Software Development Process

In software development process, there are a number of phases that the software project must pass through. As these are highly accepted development phases to be productive, we are going to apply them for our project. Note that these phases have different scales which are going to be determined by the software development approach to be employed.

There are five major phases of software development process. These are:

Communication – Requirement Analysis: In this phase, we are going to gather the requirements of the end users. We have planned to perform this by formal and informal interviews, by electronic communication with some scholars, designers and students. This information will be important to decide the features to be included in the software.

Design – Planning: We are going to prepare a "map" called software project plan that guides us through the complicated journey of the project. It defines the software engineering work by describing the technical tasks to be conducted, the risks that are likely, the resources that will be required, the work products to be produced and a work schedule.**Design – Modelling:** Models will be created to better understand software requirement and the design that will achieve those requirements.

Construction – Implementation and Testing: In this phase the code generation (both manual and automated) and testing that is required to uncover errors in the code are done.

Deployment – Integration and Maintenance: The software, as a complete entity or as a partially completed increment, is delivered to the customer who evaluates the delivered product and provides feedback based on the evaluation. For the time being, our customers are the evaluation committee and our advisors.

In addition to these framework activities, we will conduct other activities called umbrella activities. These activities are those activities applied throughout the project. Typical umbrella activities include:

- Software project tracking and control
- Risk management
- Software quality assurance
- Technical reviews
- Work product preparation and production, etc.

Several software development approaches have been used since the origin of information technology. These are:

- Waterfall: a linear framework
- Prototyping: an iterative framework
- Incremental: a combined linear – iterative framework
- Rapid application development (RAD): an iterative framework
- Extreme programming

Among these approaches we applied the **Software Prototyping Approach.** It is a development, the creation of prototypes, i.e. incomplete versions of the software program being developed. These prototypes do not have the characteristics of the finished component.

The basic principles are:

- Not a standalone, complete development methodology, but rather an approach to handle selected parts of a larger, more traditional development methodology (i.e. incremental, spiral or rapid application development (RAD))
- Attempts to reduce inherent project risk by breaking a project into smaller segments and providing more ease-of-change during the development process.
- User, in our case the evaluators of advisors, is involved throughout the development process, which increases the likelihood of user acceptance of the final implementation
- Small-scale mock – ups of the system are developed following an iterative modification process until the prototype evolves to meet the users' requirements.
- While most prototypes are developed with the expectation that they will be discarded, it is possible in some cases to evolve from prototype to working system.
- A basic understanding of the fundamental business problem is necessary to avoid solving the wrong problem.

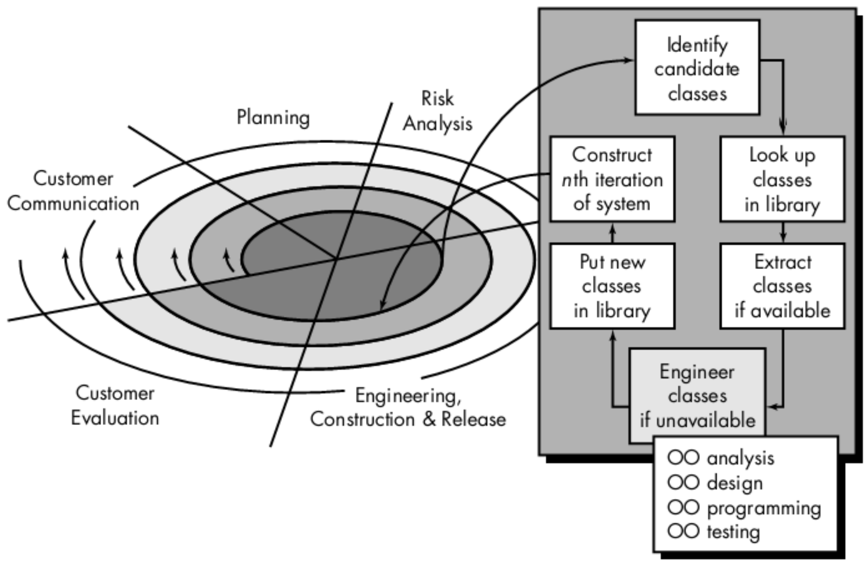
By applying the above mentioned principles, we constructed the major components and features in the first round and different improvements in each increments thereafter.

By using these methods, the construction and deployment phases were applied by using C# language in Visual Studio 2010 development environment. C# is one of the latest programming languages provided by Microsoft.

3.2. Object Oriented Paradigm

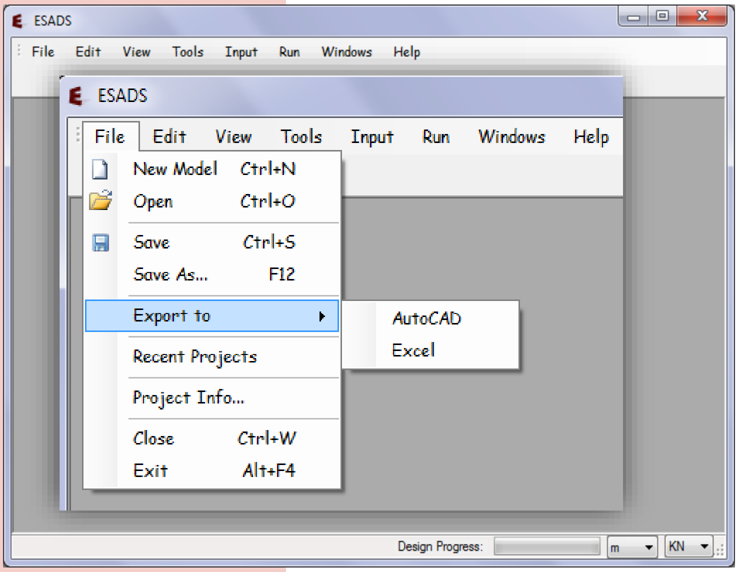
For many years, the term object oriented (OO) was used to denote a software development approach that used one of a number of object-oriented programming languages (e.g., Ada95, Java, C++, Eiffel, Smalltalk). Today, the OO paradigm encompasses a complete view of software engineering. Edward Berard notes this when he states [BER93]:

The benefits of object-oriented technology are enhanced if it is addressed early-on and throughout the software engineering process. Those considering object-oriented technology must assess its impact on the entire software engineering process. Merely employing object-oriented programming (OOP) will not yield the best results. Software engineers and their managers must consider such items as object-oriented requirements analysis (OORA), object-oriented design (OOD), object-oriented domain analysis (OODA), object-oriented database systems (OODBMS) and object-oriented computer aided software engineering (OOCASE)



The OO process moves through an evolutionary spiral that starts with customer communication. It is here that the problem domain is defined and that basic problem classes are identified. Planning and risk analysis establish a foundation for the OO project plan. The technical work associated with OO software engineering follows the iterative path shown in the shaded box. OO software engineering emphasizes reuse. Therefore, classes are "looked up" in a library (of existing OO classes) before they are built. When a class cannot be found in the library, the software engineer applies object-oriented analysis (OOA), object-oriented design (OOD), object-oriented programming (OOP), and object-oriented testing (OOT) to create the class and the objects derived from the class. The new class is then put into the library so that it may be reused in the future.

4. ESADS: Development



Outline:

- ❑ Requirements
- ❑ Object Oriented approach

In this section we would like to present the basic features of ESADS by using Use Cases and activity diagrams. The general description of the user interface is followed by the use cases is presented. The creation of objects and libraries is then discussed at last.

4.1. Requirements

The user interface of ESADS has one main window having menu strip at the top and small combo-boxes to change the units of force and length. The capabilities and requirements of in relation to general functionalities is discussed below.

Use Case: Starting a New Model

Primary Actor: User

Goal in Context: To start a new model to design either continuous beam, column, footing or slab.

Preconditions: There should be a running application of ESADS.

Scenario:

1. The user clicks 'New Model' menu item.
2. The system displays the 'New Model' dialog.
3. The user chooses the units of measurement, optionally changes the project information and chooses one of the four models.

4. The system shows the new beam, column, footing or slab dialog based on the structure type chosen.

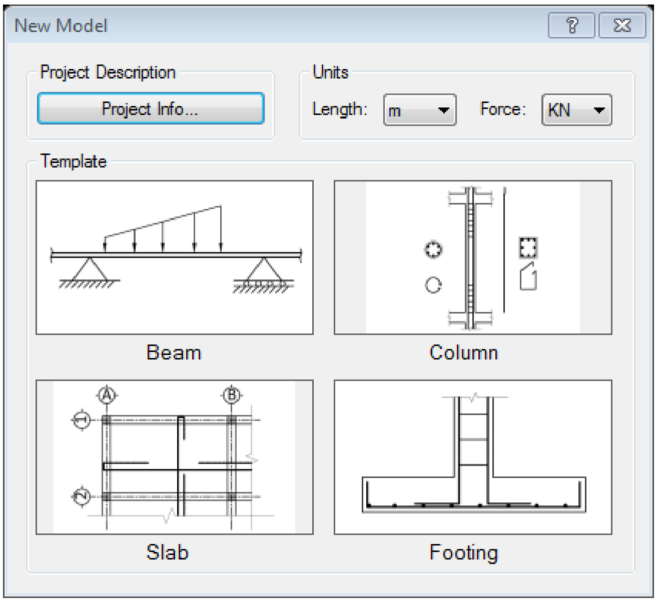


Fig 4-1: New Model Dialog

Use Case: Defining Materials

Primary Actor: User

Goal in Context: To define, modify or remove concrete and steel materials of the document.

Preconditions: There should be an active document of any model

Scenario:

1. The user clicks define material menu item or any definition button found in various other dialogs (options, new beam, new slab, new column, new footing).
2. The system displays the 'Define Material Dialog' with list of previously defined materials.
3. The system switches the list between that of concrete and steel when the user changes the type of material to be defined.
4. The system enables 'Remove' and 'Modify' button when a single element of the material list is selected. And it disables the 'Modify' button back when the user selects more than one element of the materials list.
5. The system removes all the selected materials, except those used by models, from the list of materials when the user clicks 'Remove' button.
6. The system displays 'New Material Dialog' with the corresponding material when the user clicks 'New' button.
7. The system displays 'Modify Material Dialog' filled with all the information of the currently selected material.
8. The system adds or modifies the material the material if the user accepts the dialog values of 'New' and 'Modify Material Dialog'.
9. The system resets with a warning all the material list of the currently active list when the user clicks 'Reset' button.
10. The system saves the new lists of concrete and steel to the main model document and close the dialog if the user clicks 'OK' button.

11. The system just saves the materials lists to the main model document if the user clicks ‘Apply’.
12. The system closes the dialog without saving the materials lists if the user clicks ‘Cancel’.

Exceptions:

1. Trying to accept a new material whose name already exists in the previous material list prevents the user from accepting the new material. The same is true when the user changes a material’s name to a name that an existing material has.
2. Accepting an empty material list will prevent the user from accepting the dialog values.

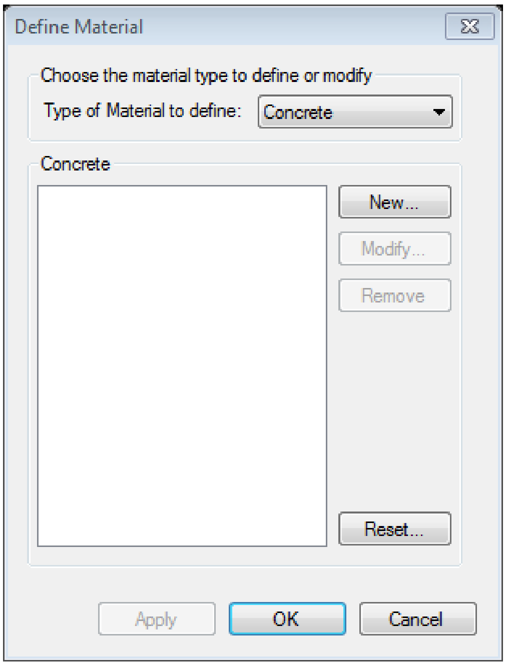


Fig 4-2: Define Material Dialog

Use Case: Creating/Modifying Material

Primary Actor: User

Goal in Context: To define new material or modify an existing one.

Preconditions: There should be an active model document.

Scenario:

1. The user clicks on ‘New’ to create new or ‘Modify’ to change the parameters of an existing one.
2. The system displays the ‘New/Modify Material’ dialog and fills it with the values if the selected material if ‘Modify’ is chosen.
3. The user chooses whether to use predefined grades or to enter all the values manually.
4. The system disables all the controls used to accept manual inputs if the user chooses to use predefined grades.
5. The system automatically fills in the corresponding values for the hidden controls whenever the user changes the grade.

6. The user may change the value of the name, unit weight, modulus of elasticity or characteristic strength values.
7. The system re-calculates the values of any parameter having unit whenever the user changes the length or force unit value.
8. The system accepts the values of the dialog and close it if the user clicks 'OK' button.
9. The system closes the dialog if the user clicks 'Cancel'.

Exceptions:

1. The user is prevented from accepting the values of the dialog if the any of the numeric values entered is zero or negative.

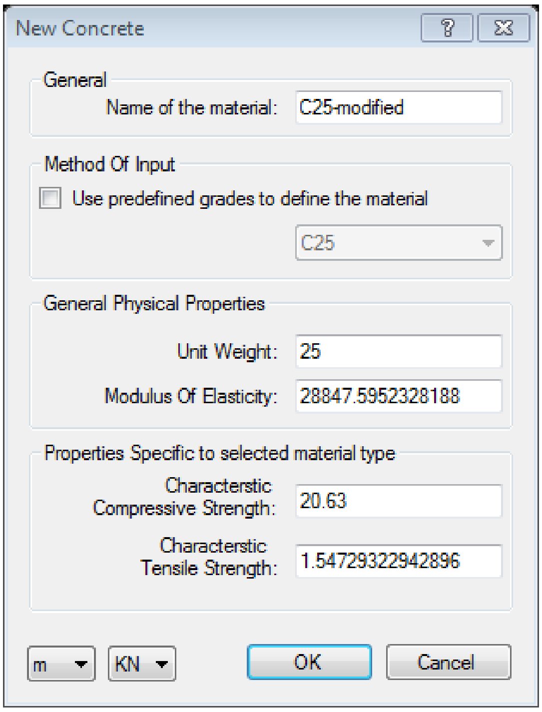


Fig 4-3: New/Modify Material Dialog

Use Case: Changing Options

Primary Actor: User

Goal in Context: To change options that apply to all models

Preconditions: There should be an active model document.

Scenario:

2. The user clicks on 'Options' menu item.
3. The system displays the 'Options' dialog filled with all previous user preferences related to the current model type of the document.
4. The user may one or more of the following:
   - May change the force and length units
   - May change the material to be used in the design of the model
   - May choose the class of work
   - May change the load factors

- May change the maximum aggregate size value
- May change the color, line type, line weight, font and visibility of different drawing layers
- May change the background color of the user window where the drawings are displayed.
- Choose whether to show the UCS icon or not
- Change the grid spacing and whether to display them or not
- Choose the color of the grid
- Choose whether to snap the grid points or not.

5. The system displays the 'Layers' dialog when the user clicks on 'Layers' button.
6. The system displays the 'Choose Color' dialog whenever the user clicks on a colored button to change the color of something.
7. The system re-calculates every value which has unit whenever the user changes the units.
8. The system saves all the preferences whenever the user clicks on 'Apply' or 'OK' buttons
9. The system closes the 'Options' dialog if the user clicks on 'Cancel" or 'OK' buttons

Exceptions:

1. The user is prevented from accepting the preferences if the value of maximum aggregate size is zero or negative.
2. The user is prevented from accepting the preferences if the value of load factors is negative.

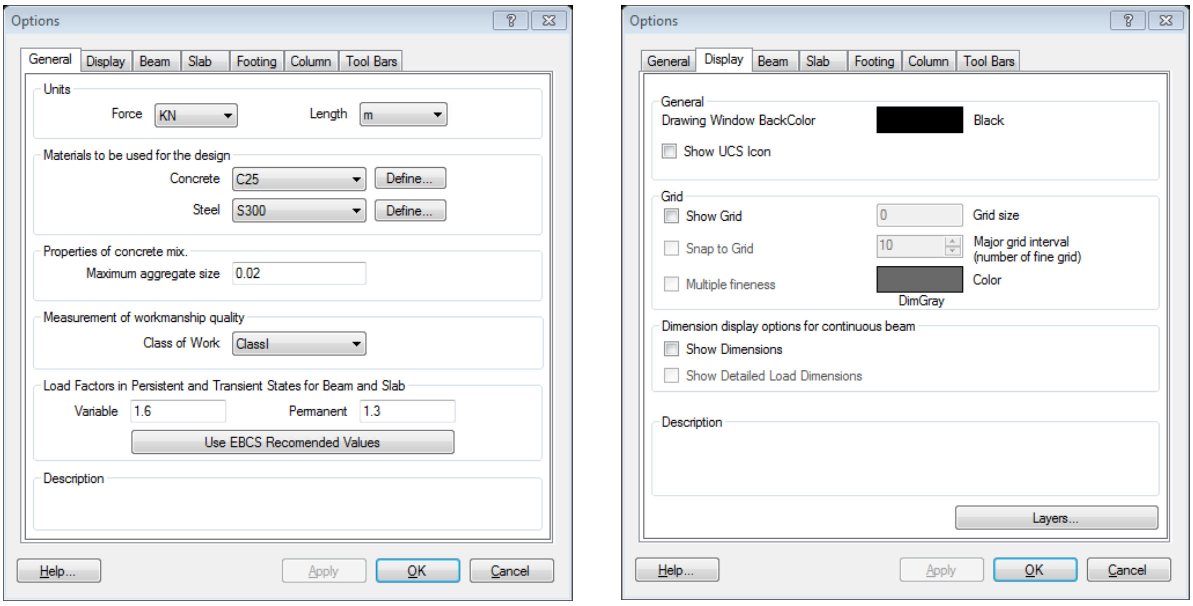


Fig 4-4: Options dialog with the General and Displays activated

4.2. ESADS: Object Oriented Approach

ESADS' code is mainly classified in to 19 libraries and assemblies. The creation of these libraries is mainly based on their degree of importance and difference with other parts. Some of the most important libraries are described below.

4.2.1. General Conventions

In this section, we will discuss the general conventions in the major activities in coding and libraries.

Namespaces:

- There should not be an ambiguity while deciding the appropriate place for a new class. If that happens, it will be an indicator of an overlap between the namespaces
- The name of a namespace should not be similar to any of the .NET's namespaces.

Classes:

- Name of every class, enumeration, delegate, structure or interface must start with 'e'. This is used to gather the classes of ESADS together at the time of coding in the 'Intellisence' of Visual Studio. (e.g. eBeam, eColumn, eLoadEventHandler etc.)

Interfaces

- Name of every interface must have an 'I' as a second character. This is used to distinguish them from the classes

4.2.2. Libraries

The following are the major libraries of ESADS.

ESADS It holds general functionalities like units conversion, documents, application. The length unit with which the system works is also found here.

ESADS.EGraphics It holds the whole drawing functionalities for all components. It has two levels, the basics and specialized. It has a functionality of drawing layers, zooming, panning etc.

It has four namespaces in it, one for each component. In these libraries the basic parts are used to build up specialized drawings. (ESADS.EGraphics.Beam, ESADS.EGraphics.Column, ESADS.EGraphics.Footing and ESADS.EGraphics.Slab)

ESADS.Mechanics It holds all the basic algorithms of the structural analysis and design. It has some special classes like those holding the functionalities of pure mathematics used in matrix analysis.

It has two namespaces under it (ESADS.Mechanics.Analysis and ESADS.Mechanics.Design) used to handle design and analysis of components separately. Each component has its own namespace in these two libraries.

ESADS.Code This library is used to hold all EBCS provisions, so that any change or checking will be easy to conduct. Most of its parts is static classes and enumerations.

ESADS.GUI This library is used to handle forms, dialog boxes and custom controls.

It is sub-divided into three namespaces (ESADS.GUI.Dialogs, ESADS.GUI.Controls and ESADS.GUI.MainForm)

The class diagram for the main libraries is shown in Appendecies.

Cable stayed bridge over the River Nile in Northern Ethiopia
Courtesy: www.google.com
In deck of the bridge is partly acting as a continuous beam over the piers and the abutments

PART II

CONTINUOUS BEAM

Chapters:

In this part, the Beam component of ESADS is fully expressed both from structural engineering and software engineering point of view. The first chapter gives a detailed introduction to the component. The main and special features of the component are presented.

Then the need for the component is described by stating the limitations of the existing design practices. Three methods are described in this chapter: manual method, spreadsheet applications and foreign softwares.

The third chapter of this part describes the structural analysis mechanism used in the internal working mechanism of the component.

Afterwards, the design mechanism of the component and how it relates to the analysis part is discussed. The detailing calculations and all code usage is also discussed in this chapter.

The last chapter of the part gives the development information of the component. Use case diagrams and activity diagrams are used to present the features and capabilities of the component. The chapter also gives brief description of classes using class diagrams.

5. Introduction to Beam

Outline:

- Overview
- Features
  - Modelling and Input
  - Member Input and Modification
  - Joint Input and Modification
  - Load Assignment and Modification
  - Running the Analysis
  - Design and Detailing Features
- Special Features

The first thing in this chapter is the brief overview of the component and what it does is given in few sentences. Then, the major features of the ESADS – Beam component is discussed in the next chapter.

5.1. Overview

The Beam component of ESADS is a component for the analysis and design of a one dimensional beam. It uses the matrix stiffness method to analyze the beam. Based on the user's preference, it designs and details the whole beam according to EBCS provisions.

5.2. Features

The beam component of ESADS is component for analysis and design of one dimensional prismatic beam. When we say one dimension we mean that every member is straight. Prismatic sections do not vary in cross-section along their length. It has three major features: The analysis, design and detailing. The scope of the component has been explained under each feature. If a beam needs to be analyzed, it should be modelled first. At his stage, ESADS is capable of modelling prismatic members of any shape and material (by providing EI manually) or rectangular concrete section (EI to be computed automatically). For the joints there are four support types (i.e. Pin, Roller, Fixed and Vertical Roller), two joint types (i.e. Hinge and Vertical Guided Roller) and tow anonymous joint types (i.e. continuous and free).

The members may be defined either graphically, or by providing the number and length of multiple members with equal length. After ending the input this way, nay of the members.

5.2.1. Modelling and Input

The first step in structural design is modelling. It is the process of representing a certain real structure with a simplified easy to manipulate object, called model. A single structure may be modelled into different models depending on the method of analysis. The model generated for matrix method structural analysis is quite different from that developed for finite element method. A continuous beam is modelled as a series of members with constant cross-section and with or without uniform loading interconnected by joints. In ESADS, the members are put in first and then the default joints may be modified by the user. Every time a beam model is created, certain inputs are expected from the user. At the very beginning the user needs to choose measurement units, concrete and steel material to be used if it is going to be designed, and further details may be modified after the creation of the beam.

In the options dialog box, the user may change wider range of inputs. This includes the units to be used, the materials used, class of work, load factors, longitudinal bar choices, stirrup bar choices, number of flexure and shear design sections, concrete cover and some further details. One of the specialties of ESADS is it allows the user either one or two bars in the flexure sections. Using more than one bar type in a sections creates an opportunity to choose the most economical bar combination, as long as the user prefers to. Furthermore, it also allows the user the way stirrups are positioned relative to the longitudinal bars.

Concerning user preferences, ESADS allows some adjustment on the graphics and overall 'On-screen-behavior'. For instance, the user may change the color of the main drawing area, the grid lines or any drawing on the drawing area to any color he/she wants. The user may choose whether or not to see the UCS icon, the grid, dimension (load dimension or at all). The user may also toggle between snap to grid and not to. Most of all the layers dialog allows the user to control every system layer used by choosing the name, color, line weight, line type, font and whether to show it on screen or not.

5.2.2. Member Input and Modification

Longitudinal member data may be input in two different ways. One of them is by just providing the length of them is by just providing the length of a member and the number of members with that length. This may be done to add a number of equal length members from scratch or to add them at the right hand side of an existing set of members. These members are added together with joints of user preference.

Alternatively, each member may be drawn with pointing devices, like mouse, by simply clicking on the drawing area consecutively. The length of the member may be simply seen from the dynamic text which changes as the pointer moves. Otherwise, the user may hit any numeric key from the keyboard which activates a text box into which the user may type in the exact length of the member and hit Enter. When the user finishes input, pressing 'Escape' terminates the dynamic member and returns the cursor to normal.

Sometimes the user may want to change the length of the member. In ESADS, this is simply done by double clicking on the member and typing in the new length, or from the properties window of the member. To remove a certain members, the user must select a series of members without gap from the right end of the beam and then hit delete key or delete menu item.

For the definition of the member cross-section, the user may right click on the member, open the properties window of the member and modify or the section menu item from input menu. When the user defines the cross-section, it may be a non-rectangular prismatic section of know EI. If this is chosen the user can only use ESADS for analysis and not for design and detailing. The design and detailing can be conducted with certain pre-conditions explained further below. Otherwise, if the user chooses to use rectangular concrete cross-section, the depth, width etc. shall be entered properly. Thereafter, ESADS will calculate the EI for that member.

5.2.3. Joint Input and Modification

When members are input in any method, the default joint is used at every place. Then the user can select any of the joints and change to any other joint. The user may even delete the joint. By deleting a joint, we mean to change the joint to the equivalent anonymous joint (continuous and free end). When a joint is deleted in the middle of the beam, it is changed to a continuous one. This means, the members joined by the joint act as monolithic parts transferring both shear and moment from one to the other. Similarly end joints are changed to free joint which means the member is overhang type.

During the definition of joints the support width shall be provided along. However, if the user is not planning to design the beam, it may be left disabled.

5.2.4. Load Assignment and Modification

One of the extraordinary features of ESADS resides at the dynamic load assignment for continuous beam. This version of ESADS supports five types of member loads and two types of joint loads. The member loads are Rectangular (Uniformly) distributed, triangular distributed, trapezoidal distributed, concentrated force and concentrated moment. Joints can be loaded with either concentrated force or moment. Any combination of the above loads may be loaded to a member as far as the capacity of the computer.

Once a load is assigned to a member, its magnitude, position and distribution length may be manipulated graphically. The small square boxes (called grip boxes) on the load enable the user to move and resize the load. By changing the sign of any load, the effect may be seen immediately on the graphics. A number may be typed any time while moving a grip box.

A number of loads may also be selected for deletion. When a member is deleted all the loads assigned to it will also vanish.

5.2.5. Running the Analysis

Once the member and joints are drawn and load assigned on the m, the user is ready to run the analysis. Before starting the analysis, the user is prompted for some preferences specific to analysis. By this time the user may choose whether to consider self-weight or not, the orientation of SFD and BMD, the way to draw the diagrams.

While the bending moment and shear force diagram is displayed, the user may right click on any member to get the summary of critical point for shear and moment. In the summary, the maximum positive and negative shear and bending moment.

5.2.6. Design and Detailing Features

After the end of the analysis the user may choose to run the design and detailing. When we are concerned with the design, some of the joint types allowed in analysis are not allowed in design and detailing. This is because they are not possible to have in concrete structures or they are too complicated to detail. Among the supports, a vertical roller and from joints hinge and vertically guided roller are not allowed in design and detailing. In addition, if any of the members in non-rectangular, the beam cannot be designed and detailed either.

Before starting to run the design, the user can change some design parameters. All the design process is held without the need for the user's interaction. At the end of the design, sections are displayed with the bars filled into them. If any section has failed during the design, the section detail is drawn with a big cross on it. This enables the user to notice the failure, see the detailed information and then take appropriate measures.

When shear sections fail, the displayed text will be a text that indicates the type of failure instead of the spacing and diameter of the shear bars. Since only two legged stirrups will be used, larger magnitude of shear may create congestion.

At the end of the design, the user will have the detail drawing at hand.

6. Existing Beam Design Practices

Outline:

- ❑ Introduction
- ❑ Manual Methods
- ❑ Spreadsheet applications
- ❑ Foreign Softwares

In this chapter, we will discuss the currently applied design trends in the construction industry of Ethiopia. This information is based on informal research and observation of actual design world.

6.1. Introduction

Different design offices employ different methods to design beams of buildings and bridges. The methods may be categorized into three groups:

- Manual methods
- Spread sheet applications
- Foreign Software

Some designers employ one method for some part of the work and some other method for another part. We will describe each, and state their limitations. The description will be from the following aspects.

- Analysis method
- Design method
- Detailing
- Compliance with building codes

6.2. Manual Method

In the academic world, anything related to analysis, design and detailing should be manual. By using manual method one can apply almost anything. The only limitation is the perfection and speed.

When the analysis of continuous beams is concerned, methods like slope-deflection, moment distribution, Kani's method, flexibility method and integration methods may be applied to get the member end forces. These forces are then used to calculate the member internal forces and hence the bending moment and shear force diagrams.

After the completion of drawing bending moment diagram, the next task is to design the beam at different sections. Design of concrete sections is conducted based on formulas. By their nature, most design methods are iterative. Due to the tediousness of the iterations, most manual designers prefer to take large number of assumptions.

Finally, the detailing is drawn based on the calculated results. The drawing may be done either with manual instruments or using CAD softwares.

Every provision of the local codes may be met due to the flexible nature of the method.

6.2.1. Limitations

The major limitation of any manual method is lack of accuracy, consistency and speed. Due to the tedious nature of design procedures, unnecessary assumptions are made. Every time the designer conducts the design of a beam, he/she is obliged to repeat so many boring steps, which by itself distracts people.

6.3. Spreadsheet Applications

The analysis of a continuous beam may be adapted on excel easily. Method of moment distribution is the easiest to adapt to spreadsheets. The member internal forces are then calculated by using the section method.

Spreadsheets are quite handy in handling beam section designs. A number of beams may be designed by just dragging an already written formula. Some sophisticated excel templates handle multiple design iterations and multiple choices between bar types. Anchorage length can be calculated and checked as well.

When we are concerned with the detailing, no excel template can generate detail drawing in any applicable form. The designer is required to draw the detail manually or using CAD softwares.

Excel templates may be developed locally that comply with the country code. Therefore, every designer may modify the template to act in accordance with their needs.

6.3.1. Limitations

The input method of spreadsheet applications is not graphical. This makes it difficult to visualize the structural mode. It also makes it difficult to detect errors and to correct them easily. The analysis results, i.e. the shear force and bending moment diagrams, cannot be presented graphically.

To proceed to the design of the beam, the analysis results must be used. This is totally done by collecting the maximum values of the diagrams manually. In the design, it is not possible to relate multiple sections of a single member. It is also quite difficult to deal with multiple row of bars in a single section.

The detail drawing cannot be drawn by these templates. Therefore, the designer is compelled to handle the output and draw the detailing. In doing so, there will be some calculation left for the draughtsman like calculation of laps, and bar cutting lengths.

6.4. Foreign Softwares

In this highly developed computer era, there are a variety of structural analysis and design softwares available commercially. Some of the common softwares seen in Ethiopia are:

All of these softwares have graphical input and output methods. They enable the definition of cross-sections used for the analysis. Shear force and bending moment diagrams may be drawn.

Design of the section is conducted based on different country codes. Some of them only calculate the area of steel while others like SAFE generate the detail drawing.

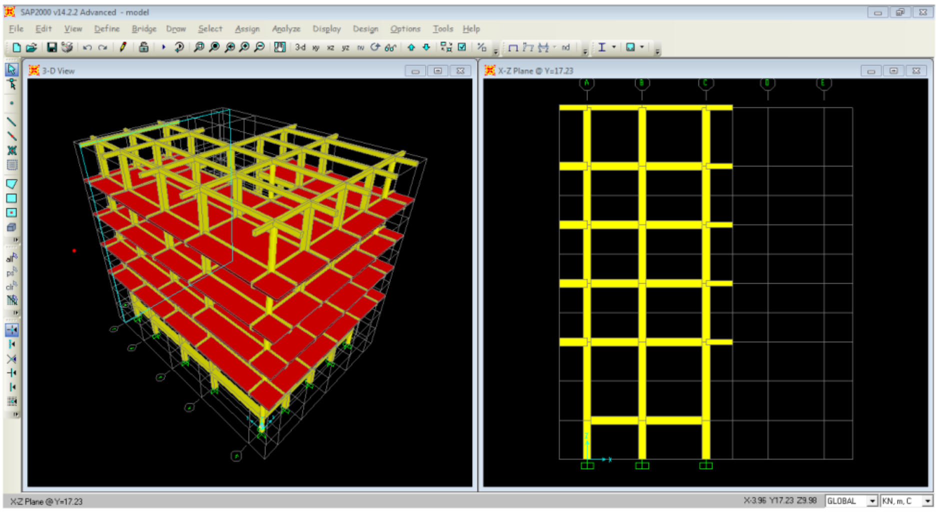
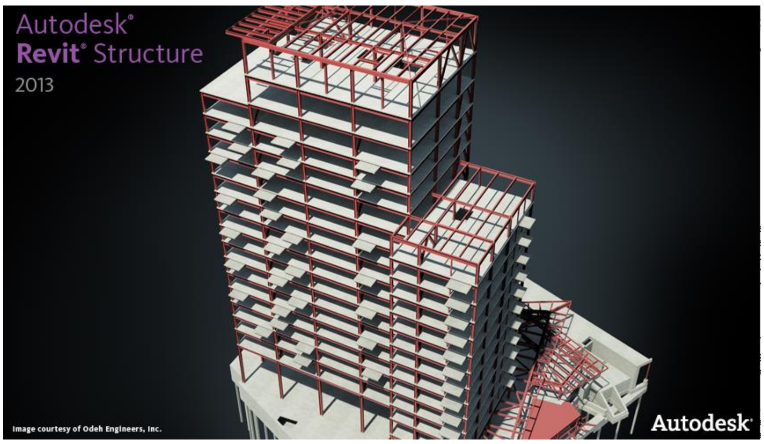


Figure 6-1: Model of a G+4 building done using SAP2000 v14.0

6.4.1. Limitations



Load input method of ETABS and SAP2000 is not flexible in terms of editing. Once a load is added it cannot be edited. If the user makes a mistake in inputting loads, he has to either remove previously added loads or cancel its effect by adding another load of equal magnitude but in opposite direction. The addition of the load is not interactive as it does not have obvious description.

SAP2000 and ETABS do not generate detail drawings even number and length of bars is not generated.

The most important drawback of using these softwares is that they cannot design using the Ethiopian Building Code Standard.

7. Beam – Structural Analysis

Outline:

- Introduction
- Matrix Stiffness Method
- Load on Members
- Application of the Stiffness Method for Beam Analysis
- Member End Forces
- Member Internal Forces

This chapter is one of the most essential chapters for the beam component. The internal working mechanism of the component is discussed for the analysis of continuous beam. It starts by listing the possible methods of beam structural analysis. Then the matrix stiffness method is discussed until the end of the chapter.

7.1. Introduction

Structures may be analyzed in one of several methods. Some of the common methods for simple structures like plane frames and beams are Kani's method, Displacement methods like slope-deflection, flexibility, moment distribution methods and stiffness method. Other higher level methods like finite element method also exist, which are beyond the scope of our level.

Some of the above methods can be adapted to computers using matrices. There are essentially tow ways that structures can be analyzed using matrix methods. According to R.C. Hibbeler, "The stiffness method can be used to analyze both statically determinate and indeterminate structures, whereas the flexibility method requires a different procedure for each of these two cases". The author continues: "Also the stiffness method yields the displacements and forces directly, whereas with the flexibility method the displacements are not obtained directly."

7.2. Matrix Stiffness Method

The stiffness method involves the division of the structure into discrete elements inter connected at certain points. The force-displacement relations are established for each element. These relationships, for the entire beam, are then grouped together into what is called a structure stiffness matrix, **K**. If the beam is loaded in between two joints, their 'effect' is transferred to the end joints. Once it is established, the unknown displacements of the nodes can be determined for any given loading on the beam. When these displacements are known, the external and internal forces in the beam can be calculated using the force-displacement relations for each member.

Identification of members and nodes is one of the first steps in the analysis. Members may be formed in a number of different ways. One may form a member between each concentrated load, end or start of distributed load, point of member cross-section change, joint or end of the beam. Alternatively, members may be formed between joints, points of cross-section change or end of beam. The former one results in higher number of members and nodes, which in turn results in larger size of structure stiffness matrix. On the other hand, the later one results in fewer number of members and joints. Having fewer number of members and joints has a significant advantage when we come to internal working resources of computer programs.

In ESADS, members are formed between joints and/or end of beam. If the user wishes to have a member with two different cross-sections, then the user would have to assign a joint type of continuous at the point where the cross-section changes.

Once the members and joints are created, then nomenclature of members, joints and joints' degree of freedom. Naming of members and joints is so easy that it may be done in any pattern. On the contrast, while naming the directions on the joints, one has to be careful in differentiating those directions that are constrained and not. Directions that are not constrained are given lower number of structural stiffness matrix number.

Each joint has at least two degree of freedom. Hinge and vertically guided roller have three degree of freedom each. This is because both of them have one component that is different on the left and right side of the joint. Hinge has different value of rotational displacement on the left side member and that of the right hand side member. In the case of vertically guided roller, vertical displacement of the joint just to the left and right of it are different.

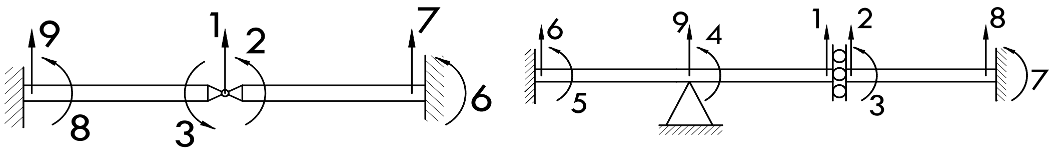
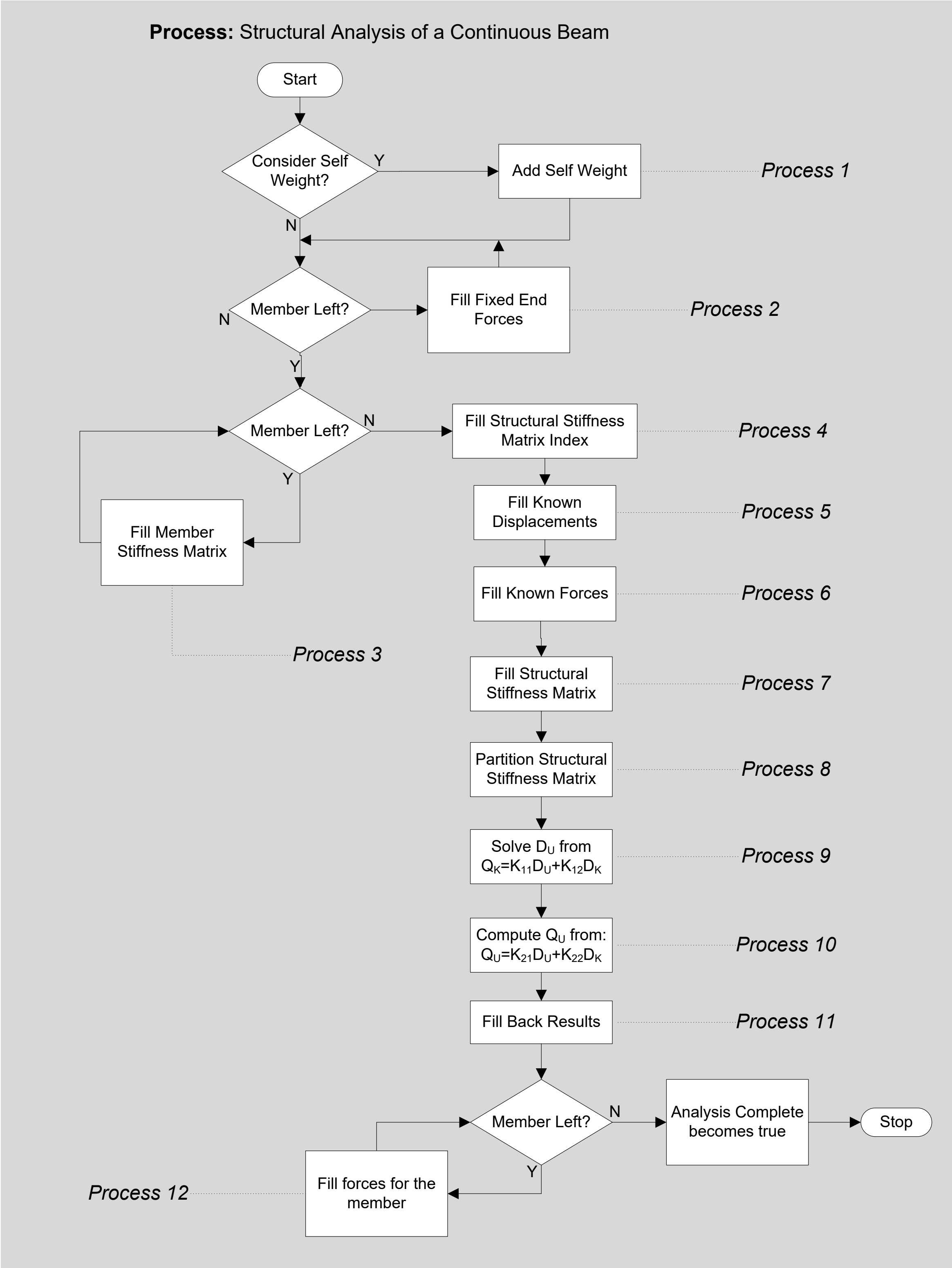


Fig. 7-1 Sample Degree of freedom nomenclature

Coordinate System: There is global and local coordinate system. The global coordinate system has its origin at the left end of the beam, and measured positive x to the right direction. The local coordinate is that start sat the left joint of each member and extends to the right end of the member.

Kinematic Indeterminacy: Once the elements and nodes have been identified, and the global coordinate system has been established, the degree of freedom for the beam and its kinematic determinacy can be determined. If we consider the effects of both bending and shear, then each node on a beam can have two degrees of freedom; namely, a vertical displacement and a rotation. The lowest code numbers will be used to identify the unknown displacements (constrained degree of freedom). This is done for convenience when partitioning the structure stiffness matrix.

Flow Chart 1: Analysis of Continuous Beam



7.2.1. Beam-Member Stiffness Matrix

Each member is required to have force-displacement relation so as to form the structure stiffness matrix. In the notation of the force-displacement relation, the left hand side of the member is called 'Near End Joint' and the other 'Far End Joint'.

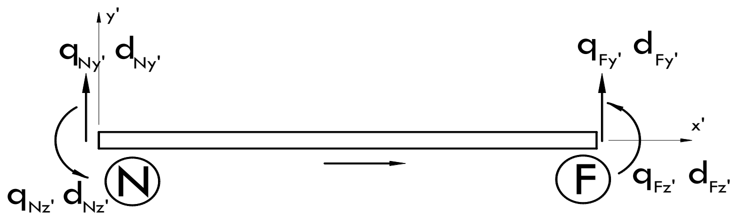


Fig 7-2 Positive Sign Convention

When the displacements in all four directions (d_{Ny}, d_{Nz}, d_{Fy}, d_{Fz}) are applied one by one while other possible displacements are prevented, the resulting shear forces and bending moments that are created are shown below.

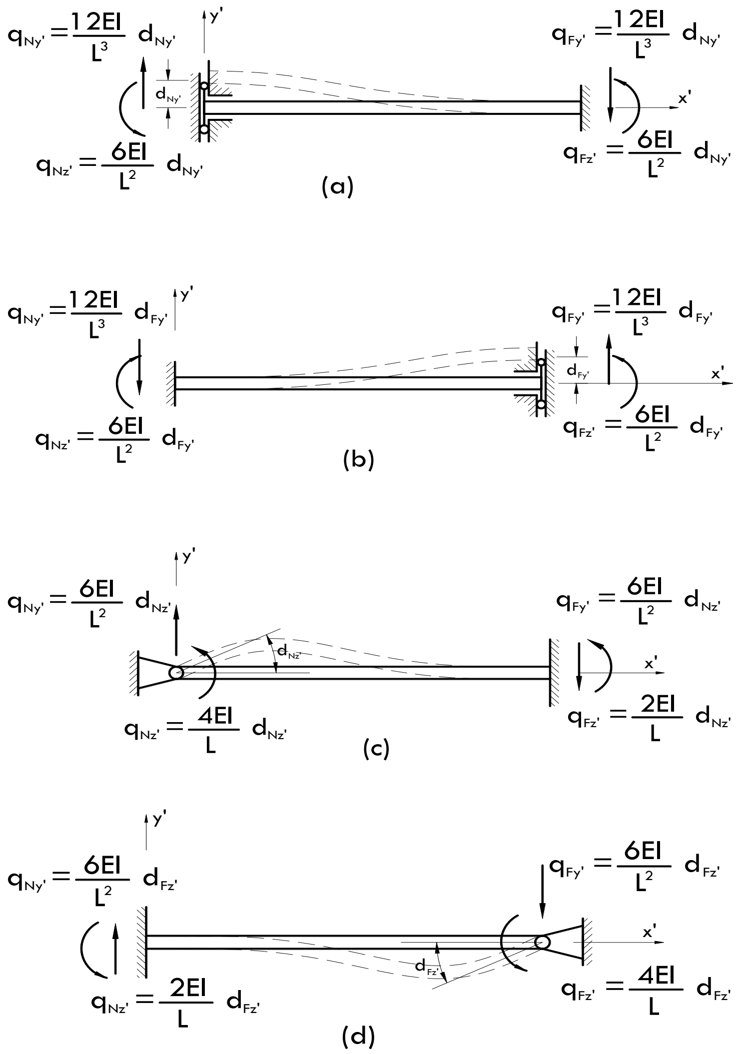


Fig 7-3: Displacements in y and z axis due to load

By superposition, if the above results in the above figures are added, the resulting four load-displacement relations for the member can be expressed in matrix from as:

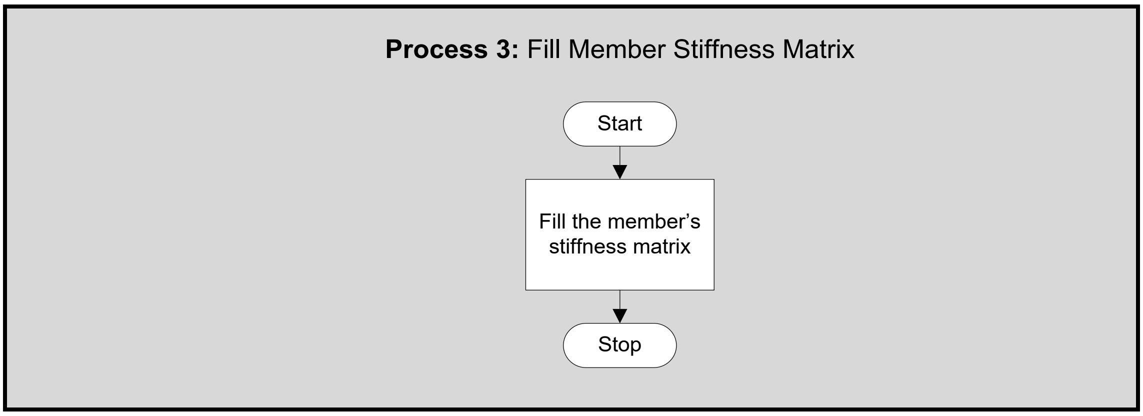
$$\begin{bmatrix} q_{Ny'} \\ q_{Nz'} \\ q_{Fy'} \\ q_{Fz'} \end{bmatrix} = \begin{array}{c} \begin{array}{cccc} \quad N_{y'} & \quad N_{z'} & \quad F_{y'} & \quad F_{z'} \end{array} \\ \begin{bmatrix} \frac{12EI}{L^3} & \frac{6EI}{L^2} & -\frac{12EI}{L^3} & \frac{EI}{L^2} \\ \frac{6EI}{L^2} & \frac{4EI}{L} & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ -\frac{12EI}{L^3} & -\frac{6EI}{L^2} & \frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ \frac{6EI}{L^2} & \frac{2EI}{L} & -\frac{6EI}{L^2} & \frac{4EI}{L} \end{bmatrix} \end{array} \begin{bmatrix} d_{Ny'} \\ d_{Nz'} \\ d_{Fy'} \\ d_{Fz'} \end{bmatrix} \qquad 7\text{-}1$$

This can be written in abbreviation as:

$$\mathbf{q} = \mathbf{kd} \qquad \textbf{7-2}$$

The numbers written above the **k** matrix are numbers given for the joint in that particular direction. This number will be used to locate the particular element on the structure stiffness matrix.

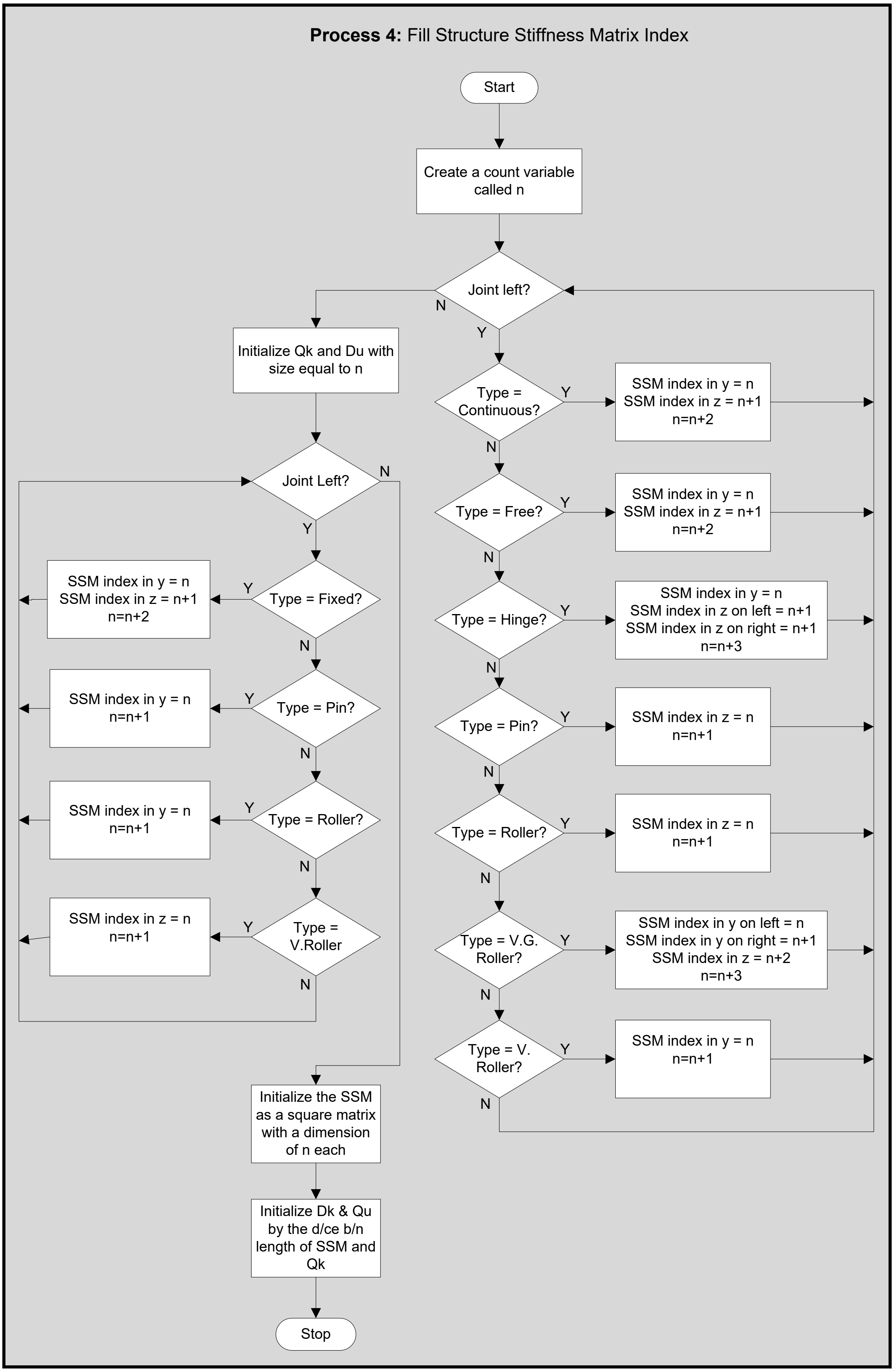
Flow Chart 2: Filling Member Stiffness matrix



7.2.2. Beam-Structure Stiffness Matrix

Once all the member stiffness matrices have been formed, they are assembled to form the beam-structure stiffness matrix. Each member stiffness matrix has numbers indicating the location of each element of the matrix in the main structure stiffness matrix. Here the rows and columns of each **k** matrix are identified by the two code numbers at the near end of the member ($N_{y'}$, $N_{z'}$) followed by those at the far end ($F_{y'}$, $F_{z'}$). When more than one number is located at the same location of the stiffness matrix, the values are algebraically added.

Flow Chart 3: Filling Structure Stiffness Matrix Index



7.3. Load on Members

Before proceeding to the application of the structure stiffness, every load should be transferred to joints. In ESADS members may be loaded in five different ways as they look for the user. Technically, there are only four types of loads managed by the basic code (Concentrated Force, Concentrated Moment, Uniformly distributed force and triangularly distributed loads). The trapezoidal load is formed from the uniformly distributed and triangularly distributed loads.

These intermediate loads may be transferred to the joints by means of fixed end forces. Consider the following uniformly distributed intermediate load:

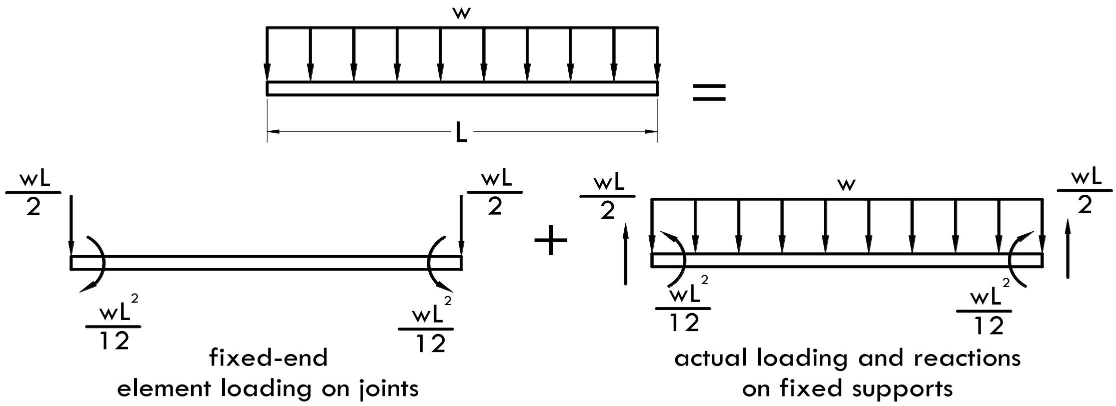
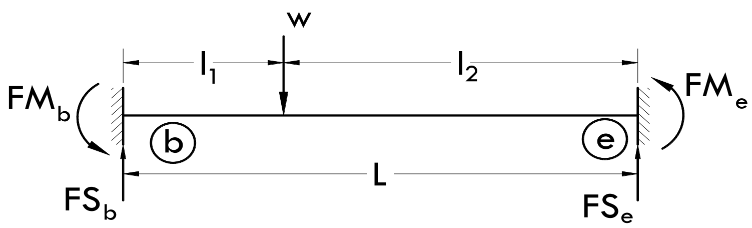


Fig 7-4: Fixed end forces for an intermediate loading

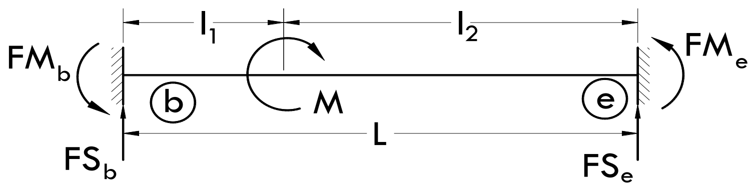
For the other loadings the following formulae were given by Aslam Kassimali in his book: "Matrix Analysis of Structures".



$$FS_b = \frac{w\, l_2^2}{L^3}(3l_1 + l_2) \qquad FS_e = \frac{wl_1^2}{L^3}(l_1 + 3l_2)$$

$$FM_b = \frac{wl_1 l_2^2}{L^2} \qquad FM_e = -\frac{wl_2 l_1^2}{L^2}$$

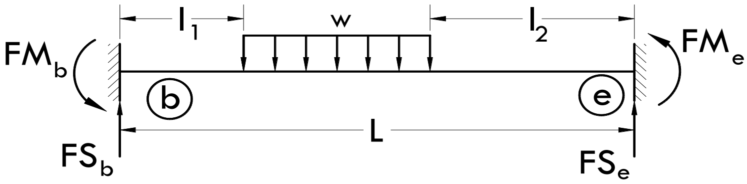
Fig 7-5: Fixed end forces for intermediate concentrated load on member



$$FS_b = -\frac{6Ml_1 l_2}{L^3} \qquad FS_e = \frac{6Ml_1 l_2}{L^3}$$

$$FM_b = \frac{Ml_2}{L^2}(l_2 - 2l_1) \qquad FM_e = \frac{Ml_1}{L^2}(l_1 - 2l_2)$$

Fig 7-6: Fixed end forces for intermediate moment loading on member



$$FS_b = \frac{wL}{2}\left[1 - \frac{l_1}{L^4}(2L^3 - 2l_1^2L + l_1^3) - \frac{l_2^3}{L_4}(2L - l_2)\right]$$

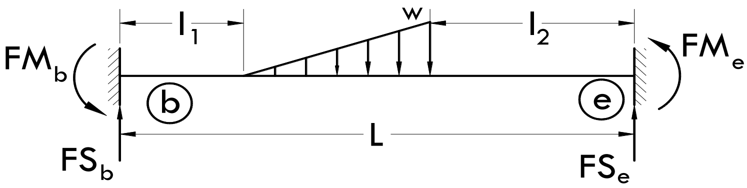
$$FM_b = \frac{wL^2}{12}\left[1 - \frac{l_1^2}{L^4}(6L^2 - 8l_1L + 3l_1^2) - \frac{l_2^3}{L^4}(4L - 3l_2)\right]$$

$$FS_e = \frac{wL}{2}\left[1 - \frac{l_2}{L^4}(2L^3 - 2l_2^2L + l_2^3) - \frac{l_1^3}{L_4}(2L - l_1)\right]$$

$$FM_e = -\frac{wL^2}{12}\left[1 - \frac{l_2^2}{L^4}(6L^2 - 8l_2L + 3l_2^2) - \frac{l_1^3}{L^4}(4L - 3l_1)\right]$$

Fig 7-7: Fixed end forces for uniformly distributed load over some length of the member

In the same book, huge formulae are given for trapezoidal loading. Since there is no need of having the trapezoidal load in our case, the below formulas are derived from those given by A. Kassimali by substituting zero for w_1 so as to get a triangular loading oriented left to right substitution of zero for w_2 leads to the orientation of right to left.



$$FS_b = \frac{w(L - l_1)^3}{60L^2}\left\{(3L + 2l_1)\left[1 + \frac{l_2}{L - l_1} + \frac{l_2^2}{(L - l_1)^2}\right] - \frac{l_2^3}{(L - l_1)^2}\left[2 + \frac{15L - 8l_2}{L - l_1}\right]\right\}$$

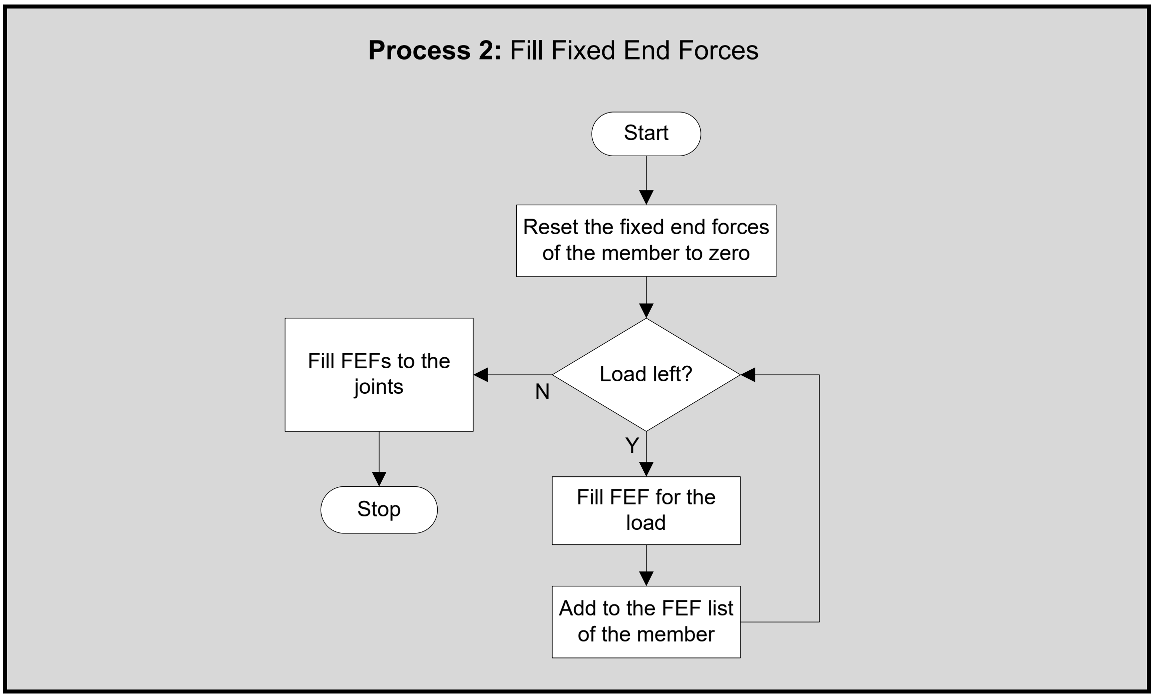
$$FS_b = \frac{w(L - l_1)^3}{60L^2}\left\{(2L + 3l_1)\left[1 + \frac{l_2}{L - l_1} + \frac{l_2^2}{(L - l_1)^2}\right] - \frac{3l_2^3}{(L - l_1)^2}\left[1 + \frac{5L - 4l_2}{L - l_1}\right]\right\}$$

$$FS_e = \frac{w}{2}(L - l_1 - l_2) - FS_b$$

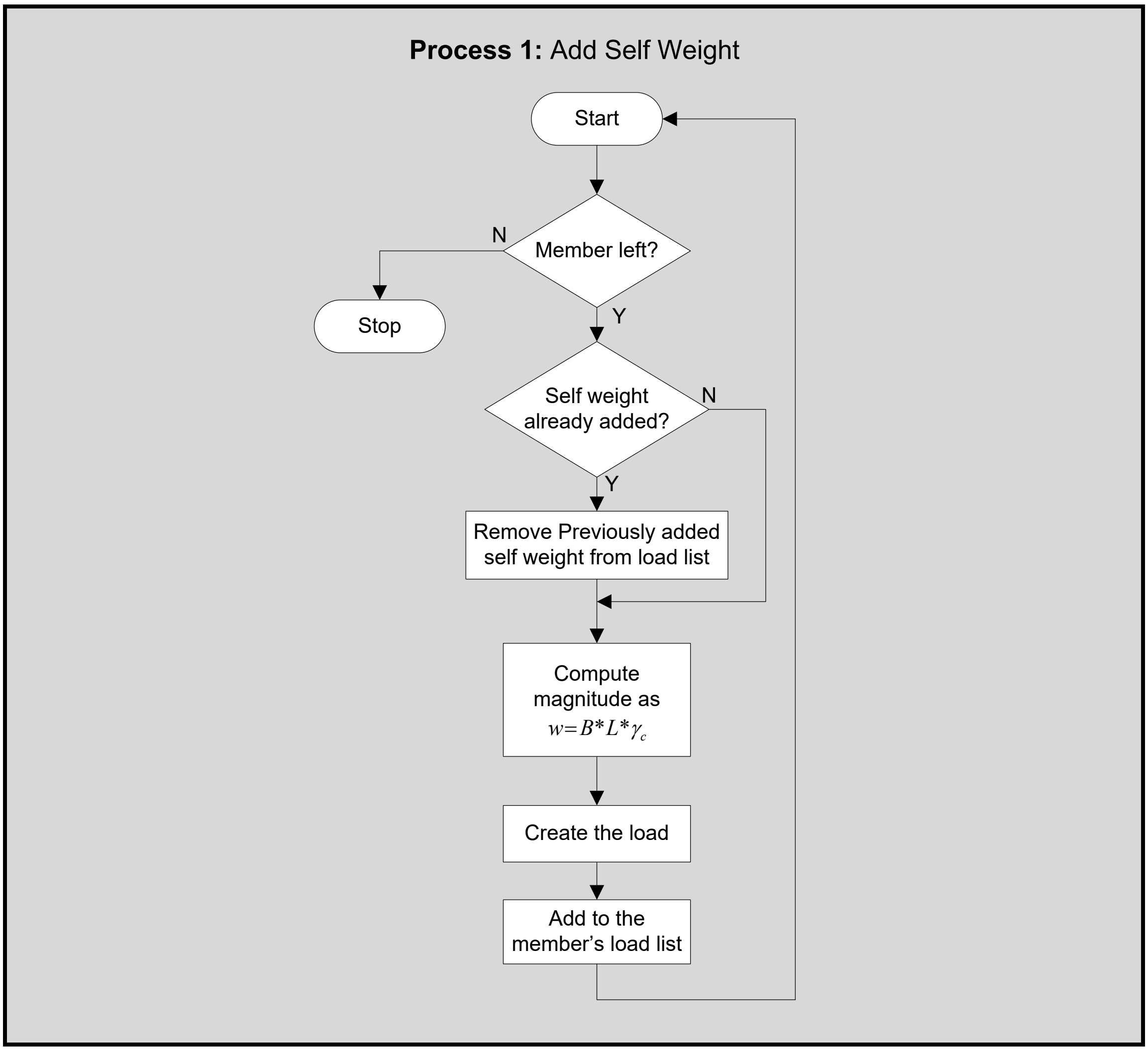
$$FM_e = \frac{L - l_1 - l_2}{6}[w(L - l_1 + 2l_2)] + FS_b(L) - FM_b$$

Fig 7-8: Fixed end reaction for triangularly loaded member for some of its length

Flow Chart 4: Filling Fixed End Forces



Flow Chart 5: Adding Self Weight



7.4. Application of the Stiffness Method for Beam Analysis

After the structure stiffness matrix has been prepared and all the fixed end forces all computed, the next thing is relating the loads at the nodes with the displacements using stiffness equation:

$$\mathbf{Q} = \mathbf{KD} \qquad \textbf{7-3}$$

Here **Q** and **D** are column matrices that represent both the known and unknown loads and displacements. Partitioning the stiffness matrix into the known and unknown elements of load and displacement; we have

$$\begin{bmatrix} Q_k \\ Q_U \end{bmatrix} = \begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix} \begin{bmatrix} D_U \\ D_K \end{bmatrix} \qquad 7\text{-}4$$

The known force and displacements are collected from joints according to the number in that particular direction.

The above relation may be expanded as:

$$Q_K = K_{11} D_U + K_{12} D_K$$

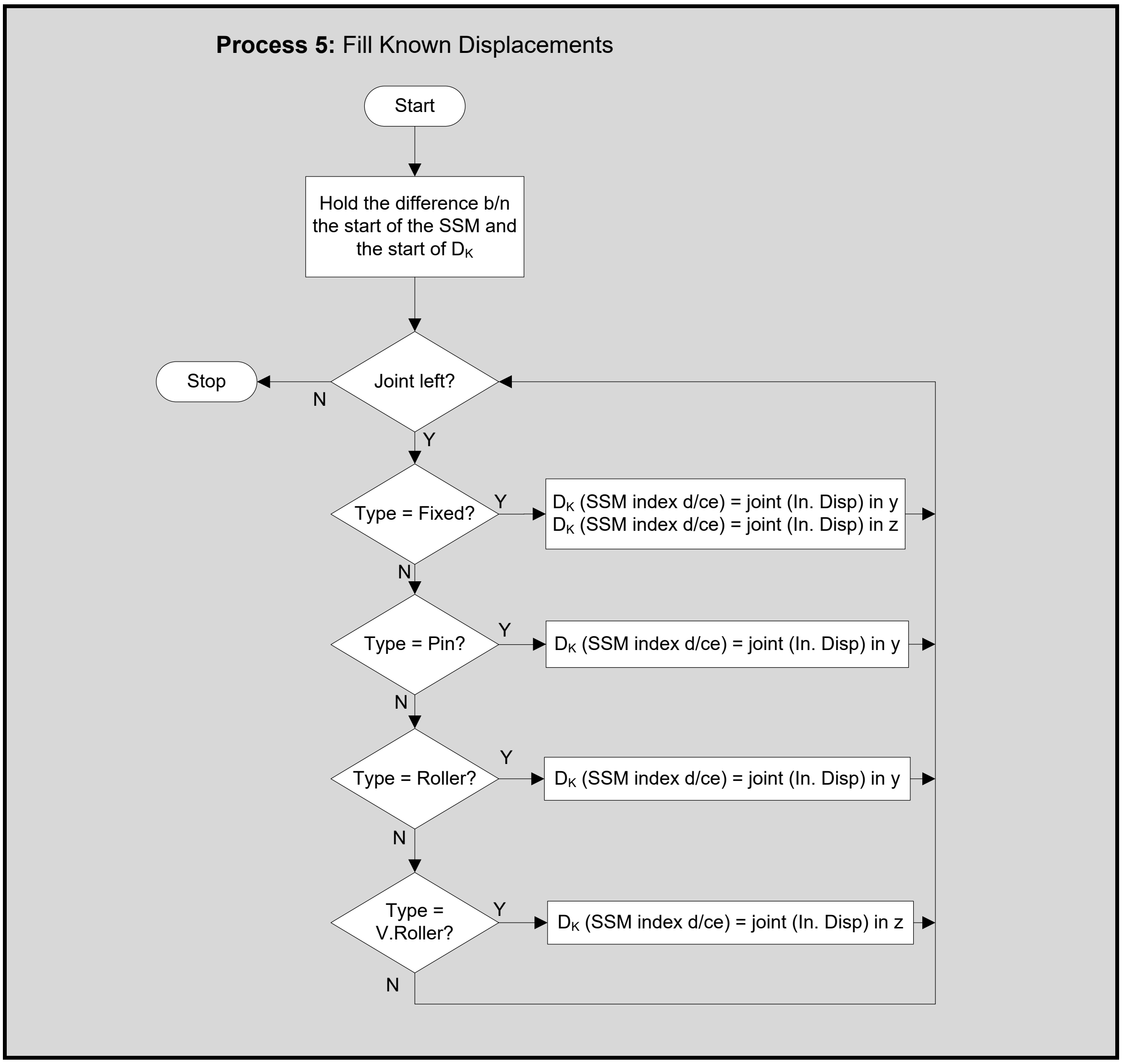
$$Q_U = K_{21}D_U + K_{22}D_K \qquad 7\text{-}5$$

From equation a, **D_u** may be determined by using any numerical method appropriate for this. ESADS uses elementary row operation on augmented matrix to change it to reduced row-echelon form.

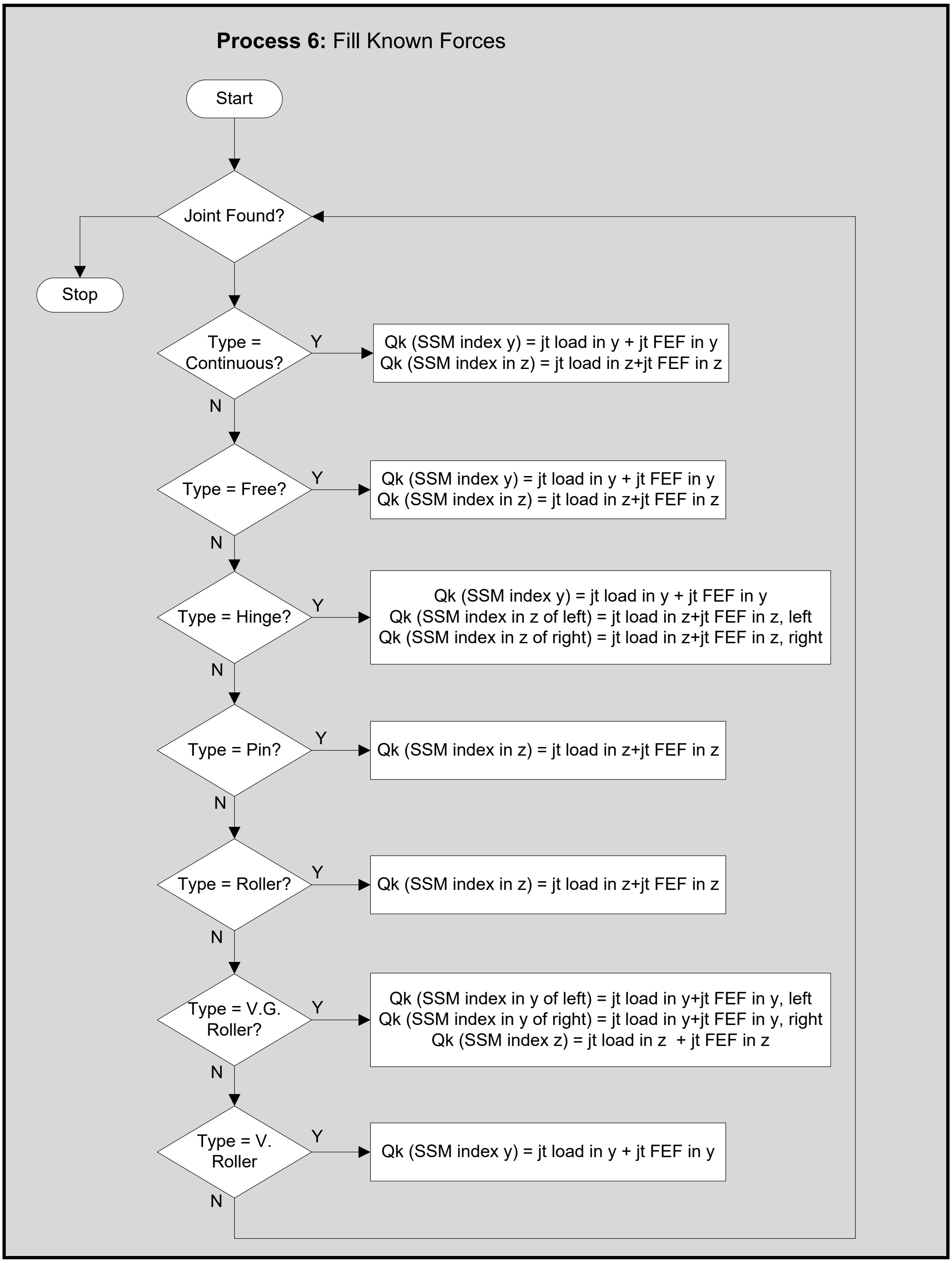
Once **D_u** is determined from equation a, **Q_u** can computed by simple matrix multiplication and addition from equation b.

Having the unknown forces and displacements, we can now fill back their values into the joints so as to get the member forces of the attached member.

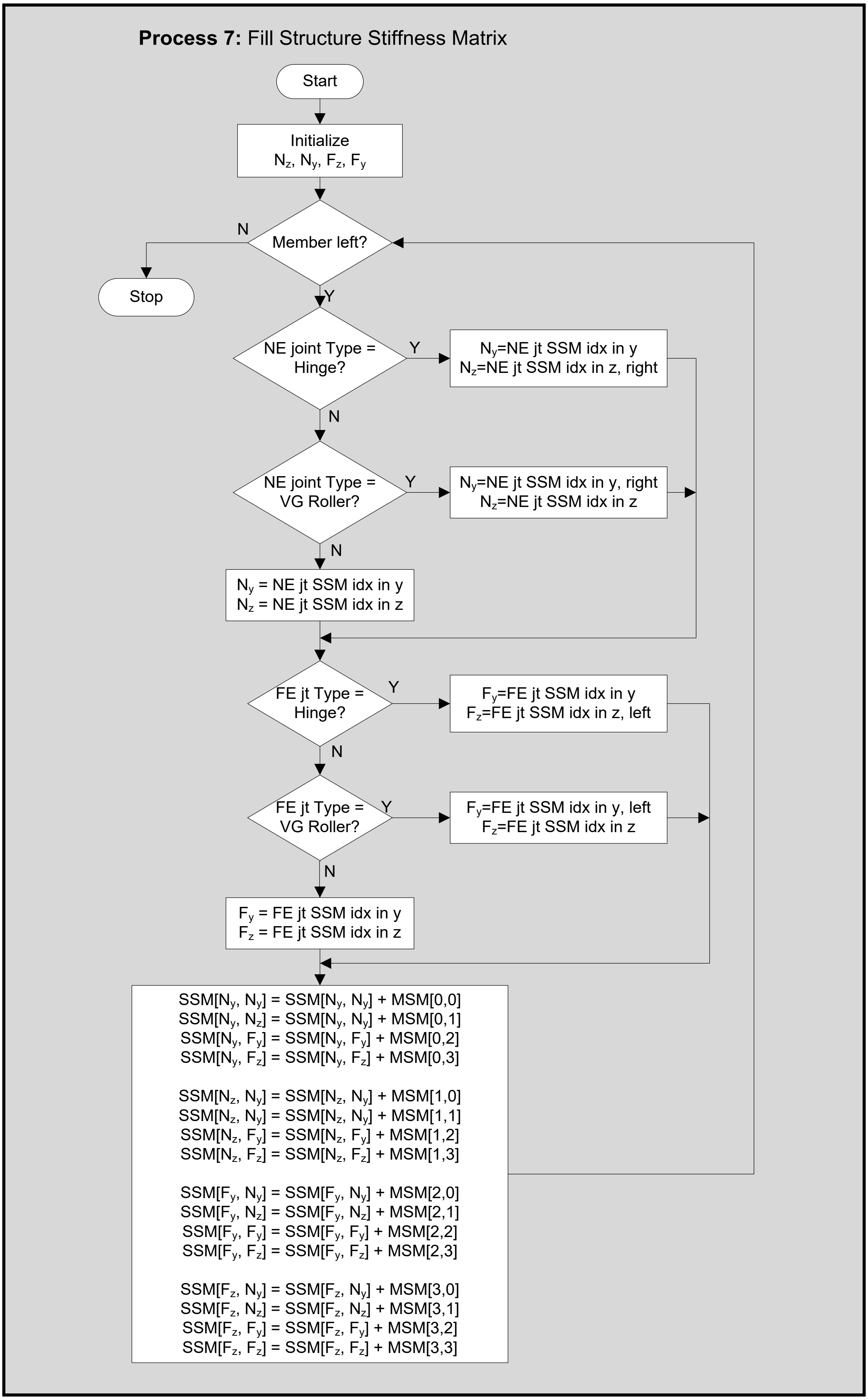
Flow Chart 6: Filling Known Displacements



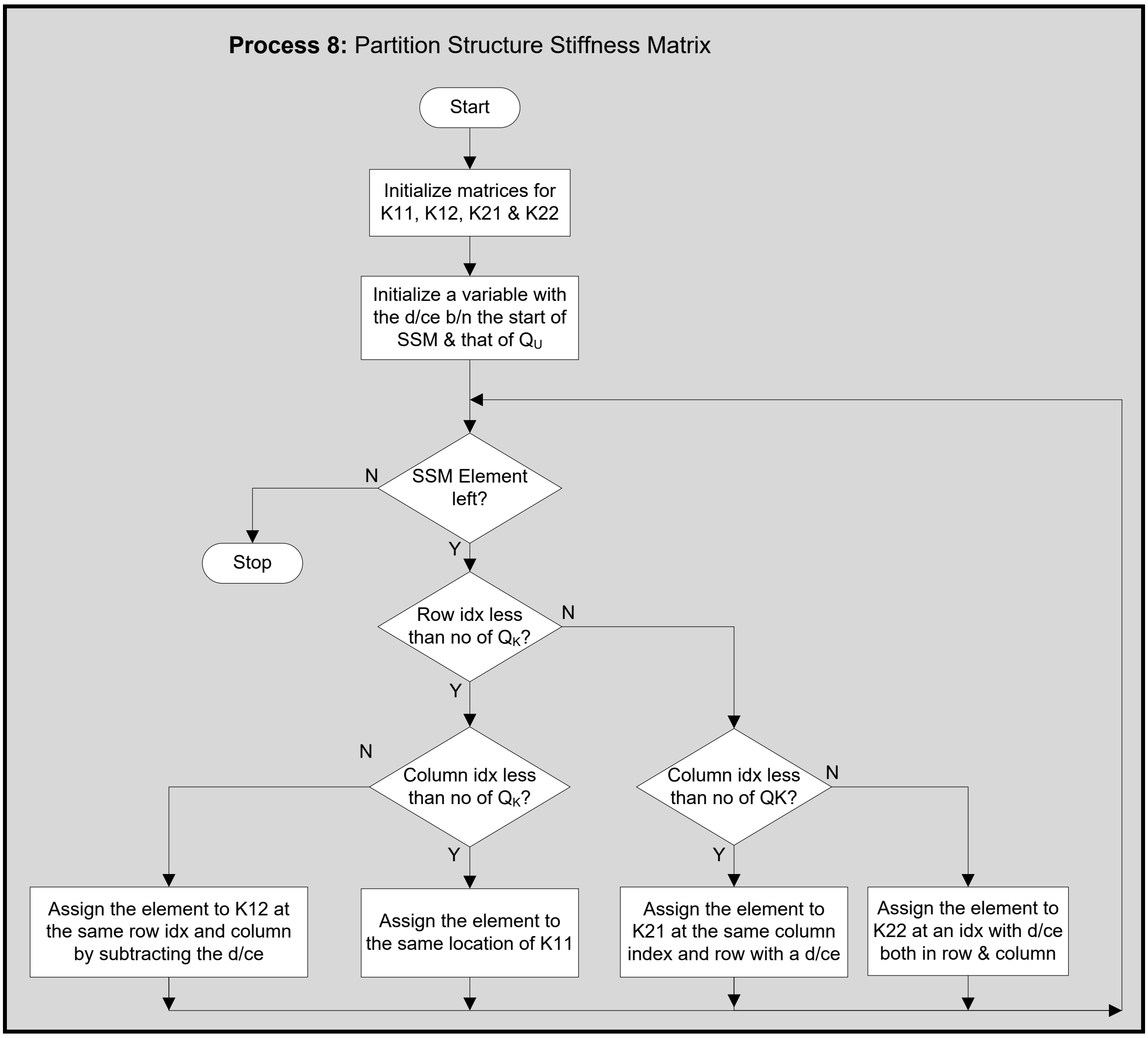
Flow Chart 7: Filling Known Forces



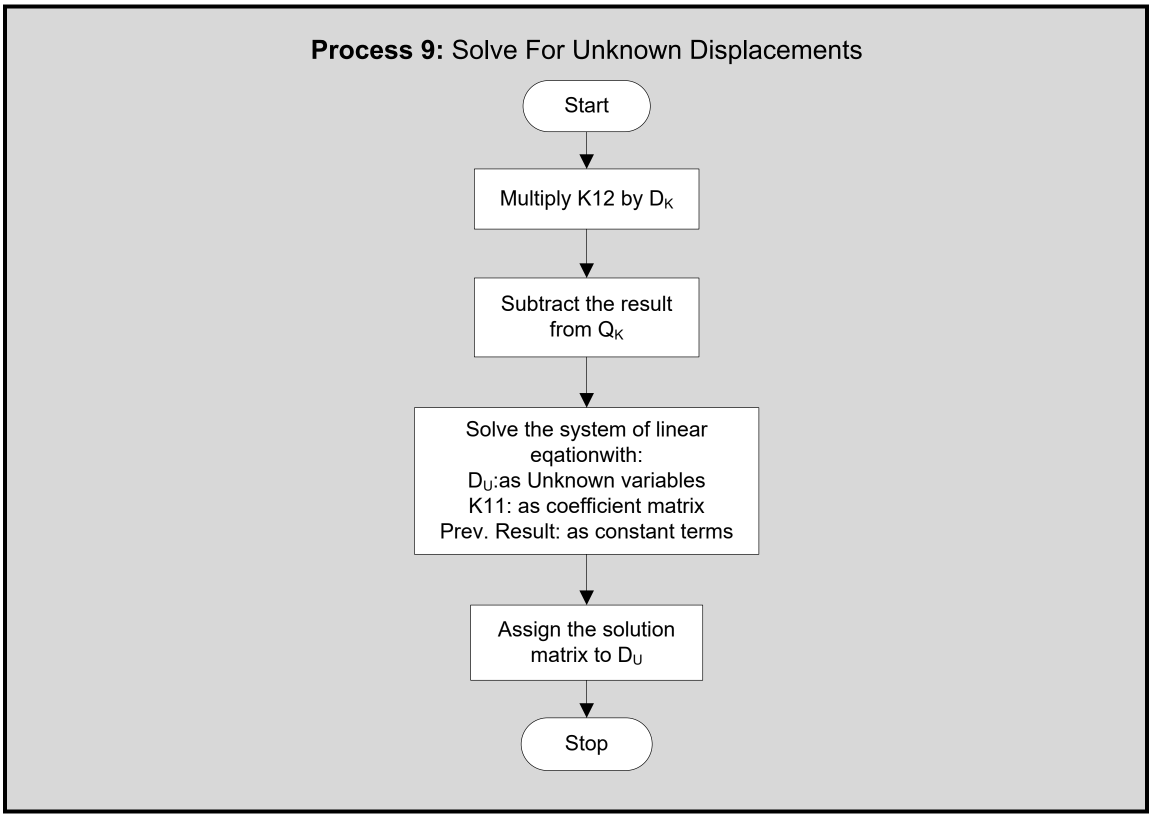
Flow Chart 8: Filling Structure Stiffness Matrix



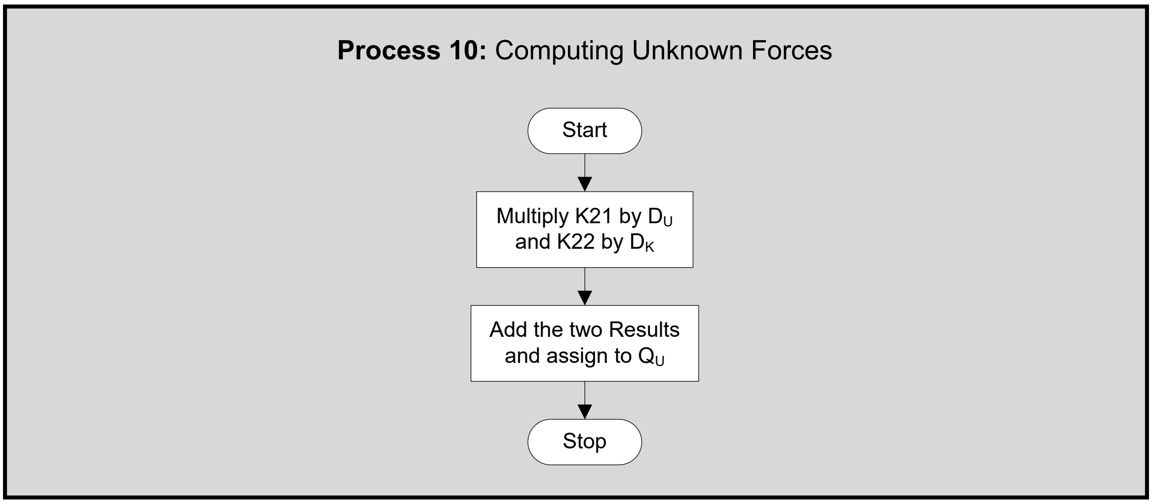
Flow Chart 9: Partitioning Structure Stiffness Matrix



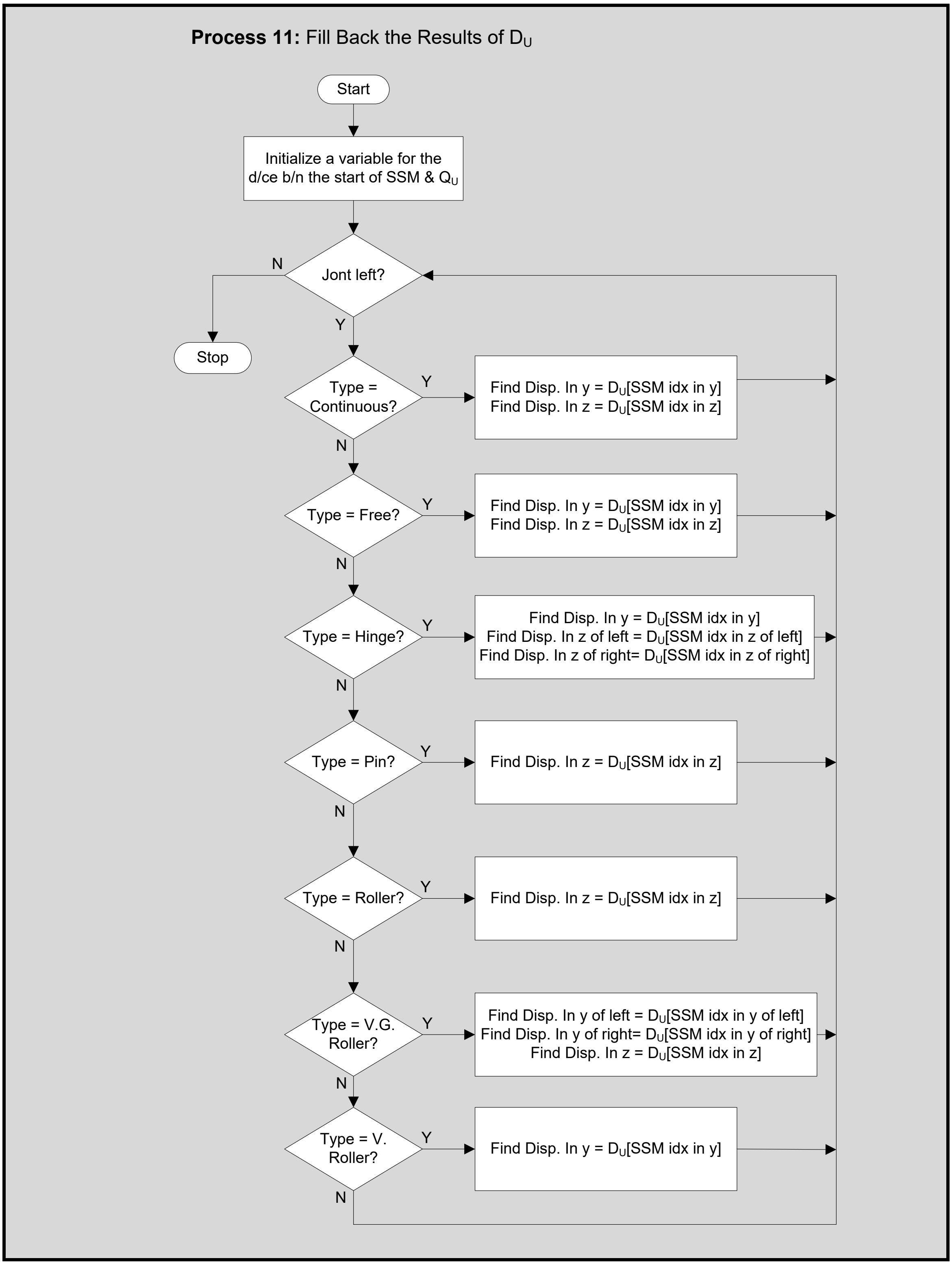
Flow Chart 10: Getting the Unknown Displacements



Flow Chart 11: Computing Unknown Forces



Flow Chart 12: Filling Back the Results of Unknown Displacements



7.5. Member End Forces

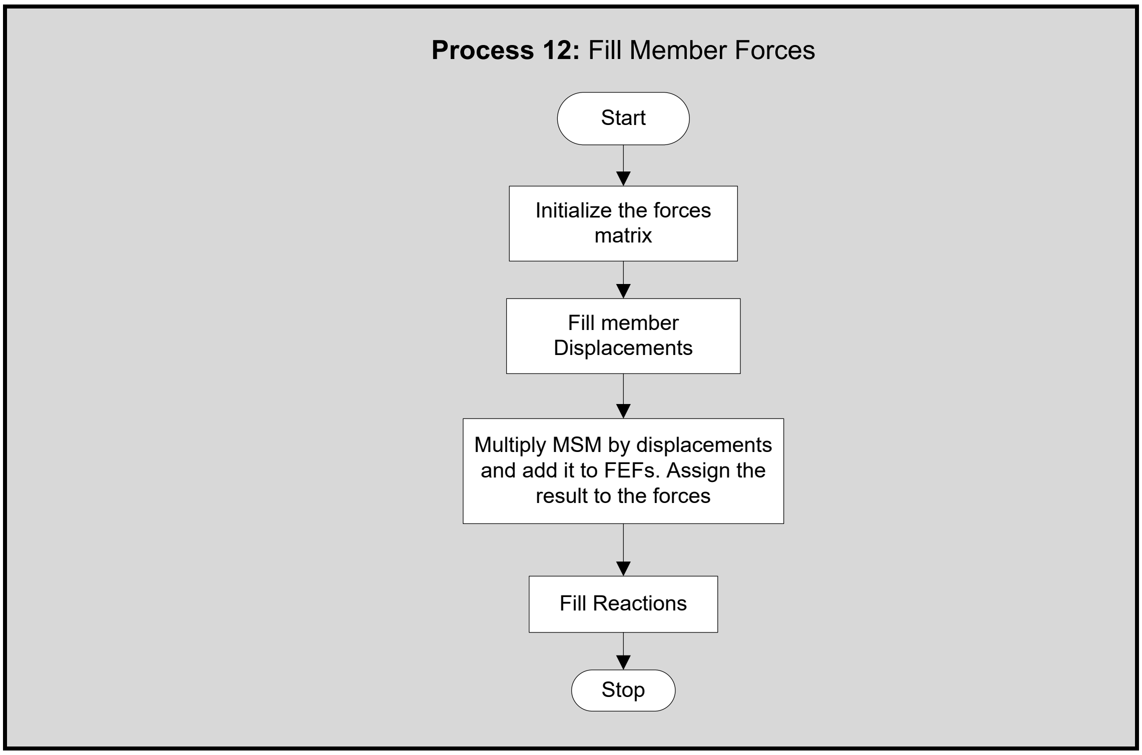
The shear force and bending moment at the ends of each member can be determined by adding on any fixed-end reactions, q_o if the element is subjected to an intermediate loading. We have:

$$q = kd + q_o \qquad 7\text{-}6$$

If the results are negative, it indicates that the loading acts in the opposite direction to that shown in fig 2.

At the end of the matrix manipulation, we have got the member end forces. However, member forces are not sufficient to make use of the analysis results. Therefore, below it is explained the way to get shear or moment at any point on the member and finding the extreme values for the member.

Flow Chart 13: Filling the End Forces of Members



7.6. Member Internal Forces

A member may be loaded with unlimited number of loads. Once the member end forces are known, the internal forces at any particular location on the member may be computed by using simple equilibrium.

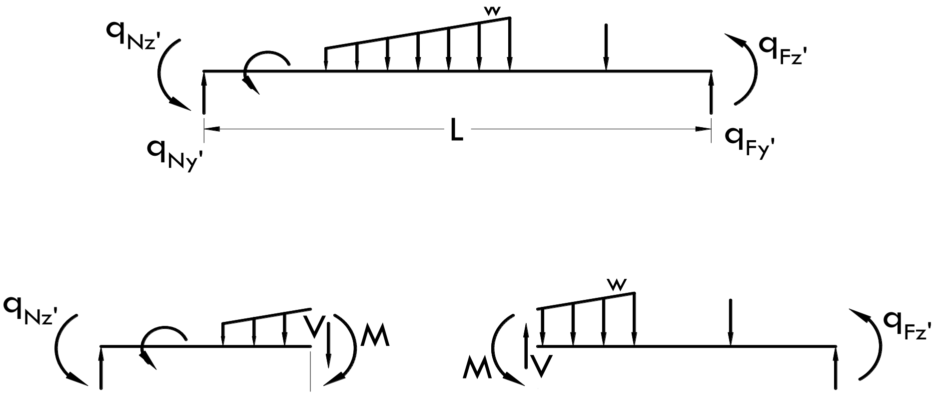


Fig 7-9: Internal forces of member

Extreme Forces in a Member: Extreme values of shear and moment are very essential, especially, if the beam is going to be designed. There are many approaches towards finding the extreme values along a member. If the load functions had been a single mathematical expressions as a function of the x-coordinate, the extreme values could have been determined by using differentiation. Even if the loads cannot be expressed as one function, some of them may be represented by polynomial functions with a maximum of degree one. This means we can use differentiation on the basis of dividing the member interval over which all the loads may be expressed as one function.

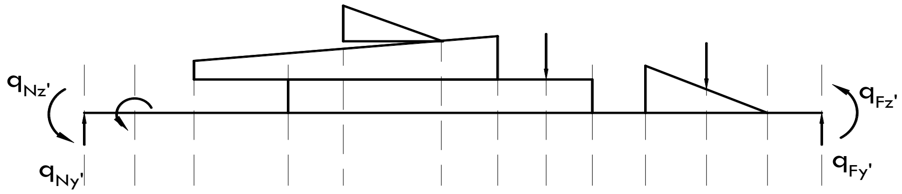


Fig 7-10: Intervals of loading which can be represented by a single polynomial expression.

As we can see from the figure, there is no possibility of having loading of higher degree than linear. Linear loading results in quadratic shear and degree three moment functions. The concentrated forces and moments will not be a problem as they will be dealt as boundary conditions for the intervals. By finding the extreme shear force and bending moment for the intervals, the shear force and bending moment for the member may be determined by simple comparison.

Consider a single interval anywhere on the member over which there is no abrupt change in loading. Then, the loading over the interval can only be one of three cases: no load, uniform load, linearly varying load. Let us consider the linear case since higher degree polynomials of lower degree. Shear force is the integration of load, whereas bending moment is the integration of the shear force as a function of the distance over the member.

The general form of a degree three polynomial is:

$$f(x) = ax^3 + bx^2 + cx + d \qquad 7\text{-}7$$

Having four points that lie on the function, the values of the coefficients may be found by solving the formed system of linear equations. If the four points are: (x_1, y_1), (x_2, y_2), (x_3, y_3) and (x_4, y_4).

$$y_1 = ax_1^3 + bx_1^2 + cx_1 + d \qquad 7\text{-}8$$

$$y_2 = ax_2^3 + bx_2^2 + cx_2 + d \qquad 7\text{-}9$$

$$y_3 = ax_3^3 + bx_3^2 + cx_3 + d \qquad 7\text{-}10$$

$$y_4 = ax_4^3 + bx_4^2 + cx_4 + d \qquad 7\text{-}11$$

Using elementary row operations, the values of a, b, c and d are computed. From these values, we may know for sure the degree of the polynomial. If the value of 'a' is zero while that of 'b' is different from zero, then the function is a quadratic function. And if the value of both 'a' and 'b' is zero while 'c' is different from zero that means the function is linear.

Having the polynomial expression, one can find the extreme values by differentiating the function. If the x value, are taken at the boundary.

To apply these methods to find extreme values of shear and moment, we have the interval. Taking the boundary values as two points on the function, we need two more points. Taking the two points that divide the interval into three equal parts will be fair assumption. Now we have four x value where we need the value of the shear or moment. To find that, we can use the methods already devised before: using equilibrium.

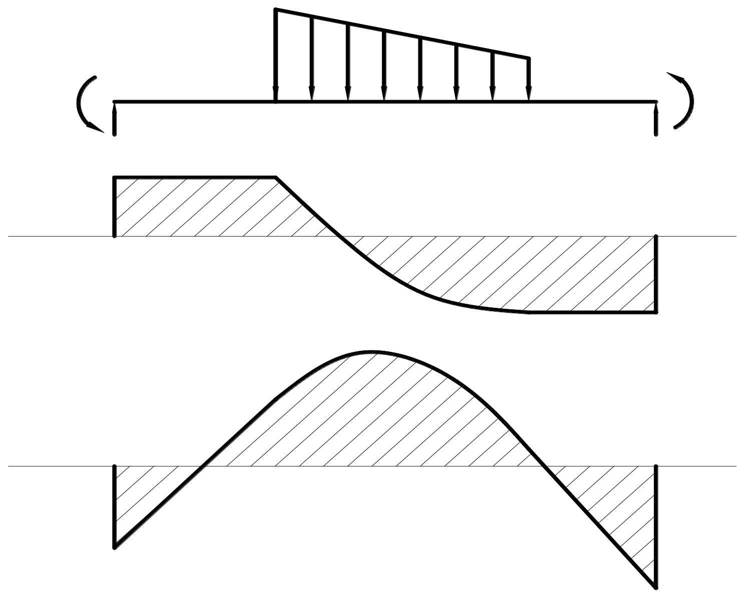


Fig 7-11: Bending Moment and Shear Force diagrams

8. Design of RC Continuous Beam

Outline:

- Use of analysis output for design
- Design of beam for flexure
- Design of beam section for shear
- Design Section Optimization
- Detailing of Continuous Beam

In this chapter, we are going to discuss the use of analysis output for design of the beam up to the detailing. Since the scope of the design part is narrower than that of the analysis part, the further requirements are also stated here. After deciding where to design for flexure and shear on the BMD and SFD, the next step is to design each section for their design moment and shear, respectively. Finally, optimization and length calculation of reinforcements is discussed.

8.1. Use of Analysis Output for Design

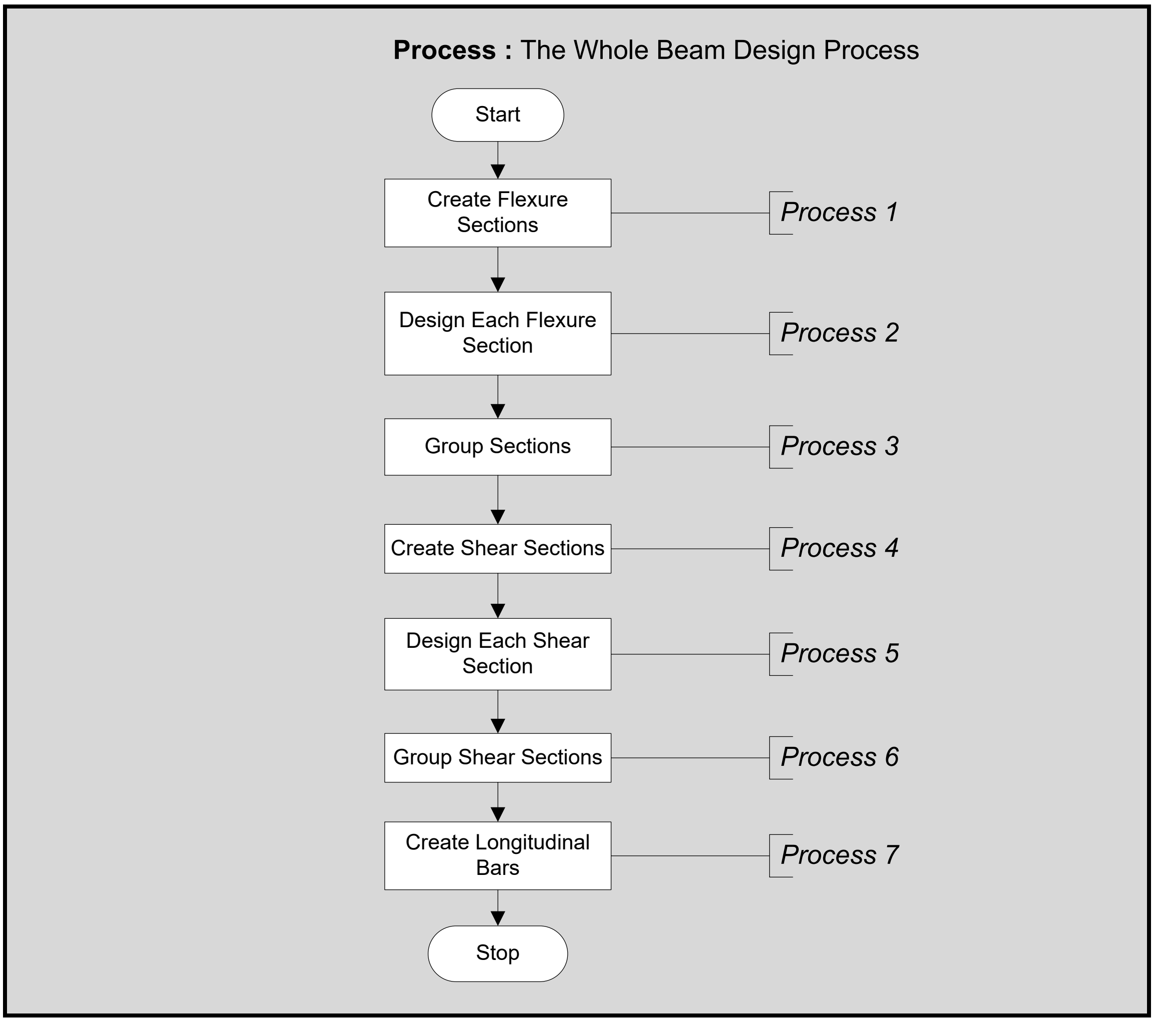
In the previous chapter, we have discussed the working mechanism of ESADS to analyze a continuous beam. Now we are going to discuss how ESADS uses the bending moment and shear force diagrams to design the beam.

By their nature, the diagrams usually vary with the length of the beam. However, it is not possible to design for every value of the diagram. This is because the final design output would be non-realistic to work with. Designing the beam for maximum values of the diagrams is, therefore, much wiser way of design.

Most of the time, building frames are loaded with distributed gravity loads not varying between adjacent spans. This type of loading usually results in a bending moment with two negative moments at each end (support moments) and a positive moment somewhere in the middle. Then the member is designed for the two support moments and the positive moment in the span. In case one of the three moments (the two support and one span moment) ceases to exist, as in the case of end span lightly loaded span between highly loaded spans, the extension of the remaining section is altered accordingly.

ESADS uses the principle of three moments per member to design the beam. If the defined load results in highly varied BMD, it takes the two support moments and the maximum of all the intermediate positive moments for the span. For end spans, it takes only one support moment and extends the span section up to the end of the member where the moment is zero. Similarly, it takes only one section for simply supported and cantilever spans. In general, ESADS uses a minimum of one and a maximum of three moment values per member to execute the design for flexure.

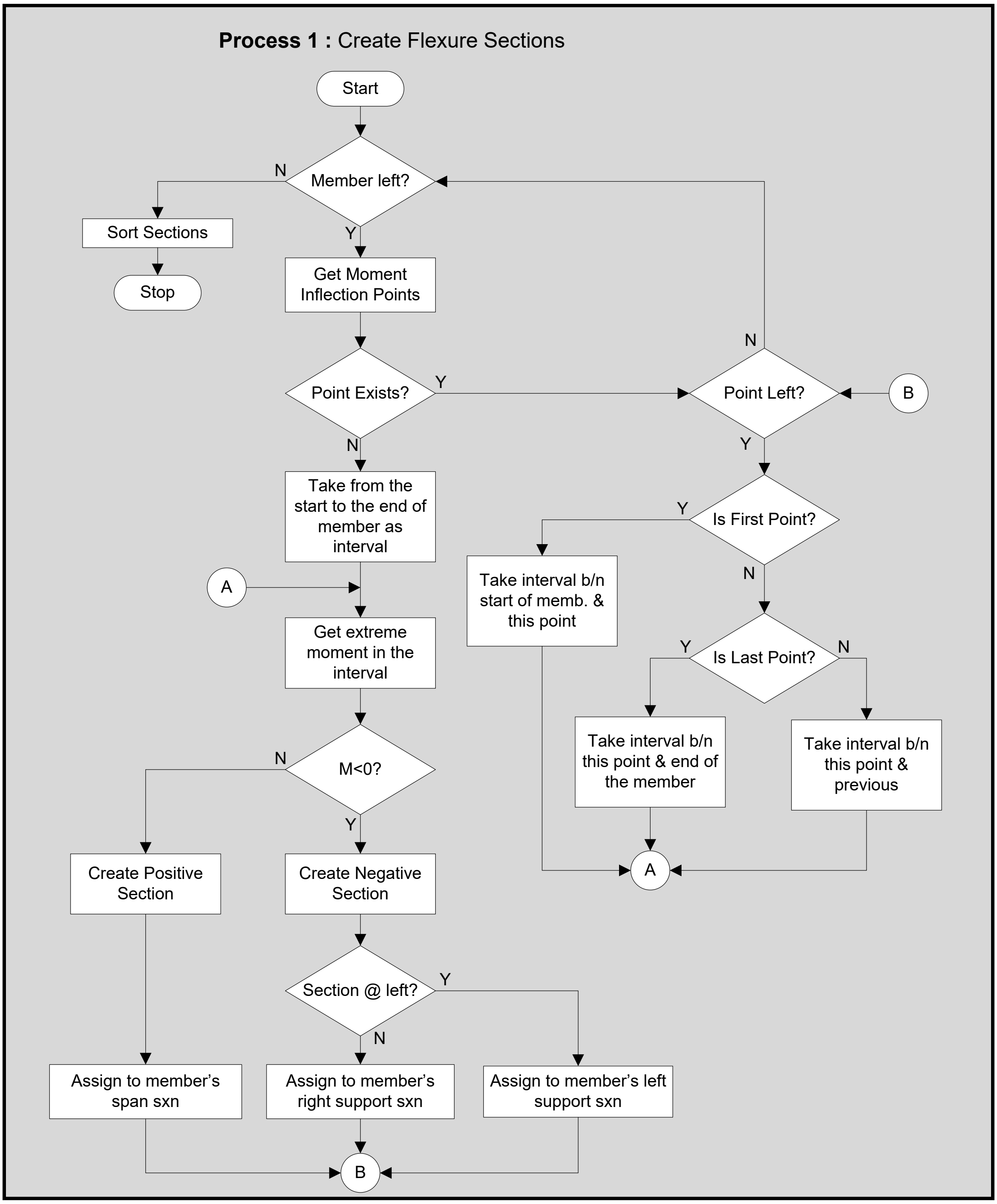
Flow Chart 14: Design of Beam – Overall Design



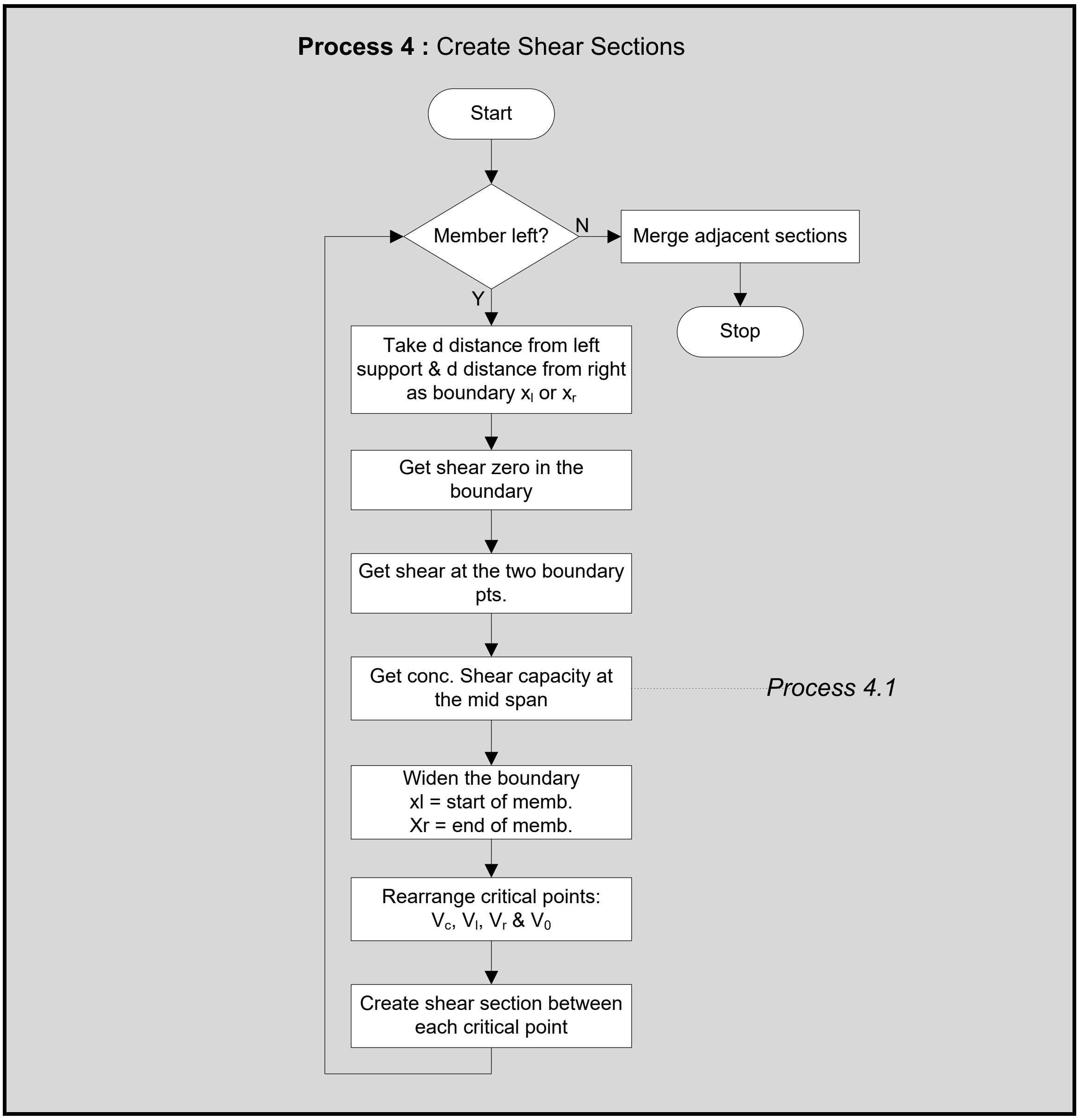
Concerning the selection of shear force values from the SFD, the design value of the shear force is taken starting from a distance equal to the effective depth, d, at that section. This is explained in detail later with shear design. The SFD of the common building frame beam members varies linearly with the length. It usually vanishes around the middle of the span. Therefore, the shear capacity of the pure concrete section lies somewhere between the shear at the supports and the point of zero shear. The distance between each critical points (at d distance from supports and where the shear is equal to the concrete's shear capacity) is then designed with the maximum shear in the interval. To earn a better economy, the designer may take finer interval division so as to avoid provision of larger reinforcement at locations with smaller value of shear just because both of the m are included in the same section.

By default, ESADS designs beams for shear by taking shear values at d distance from the supports and the concrete's shear capacity as the critical shear values to create the shear sections. Each interval is then designed for the maximum value of shear from the interval.

Flow Chart 15: Creation of Flexure Sections



Flow Chart 18: Create Shear Sections



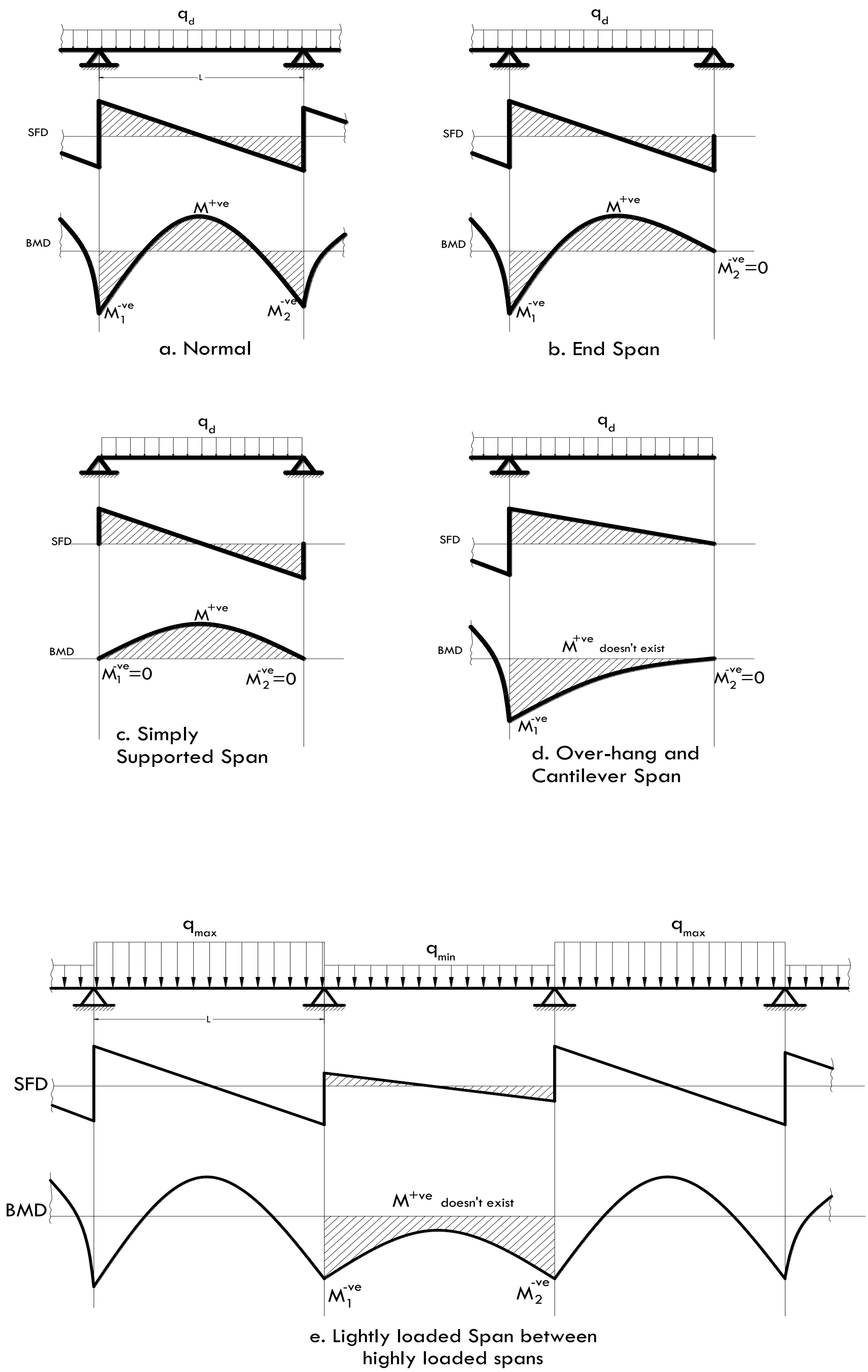
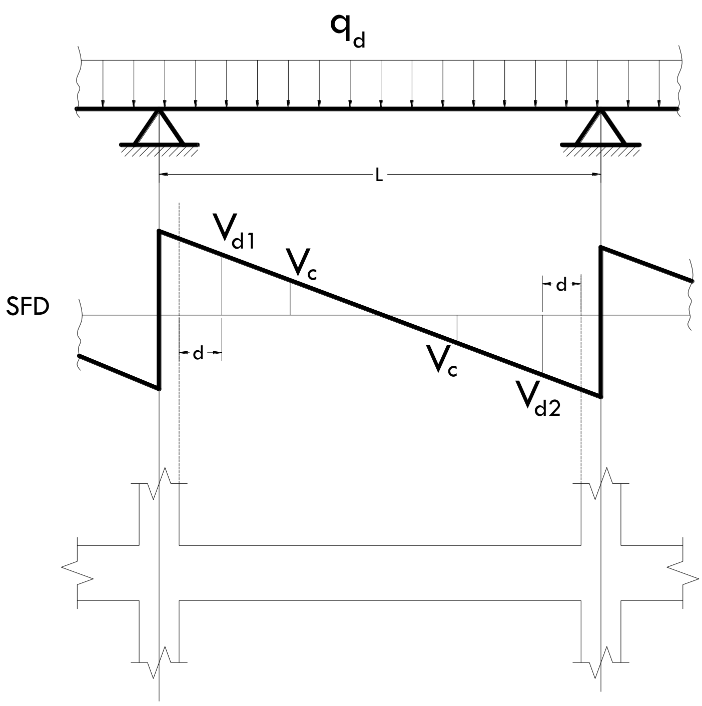
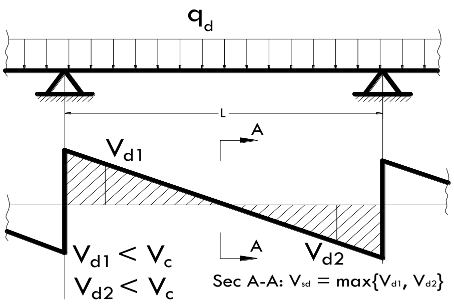


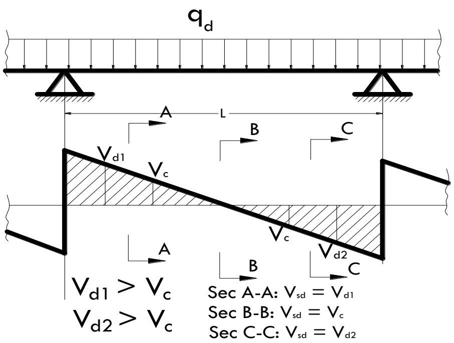
Fig 8-1: Moment diagram possibilities of common building beams



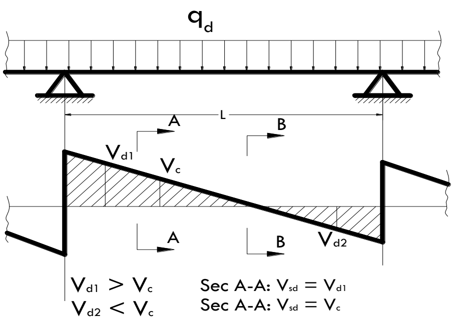
Critical Shear Values for Common Building Frame Beam



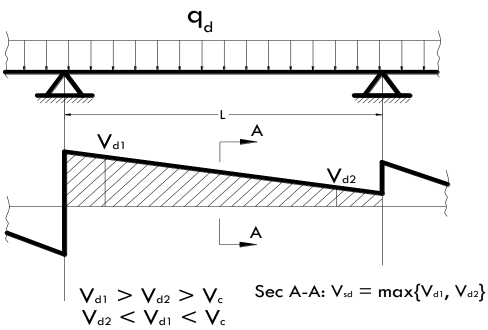
a. Single Shear Section



b. Three Shear Section



c. Two Shear Sections



d. Single Shear Section

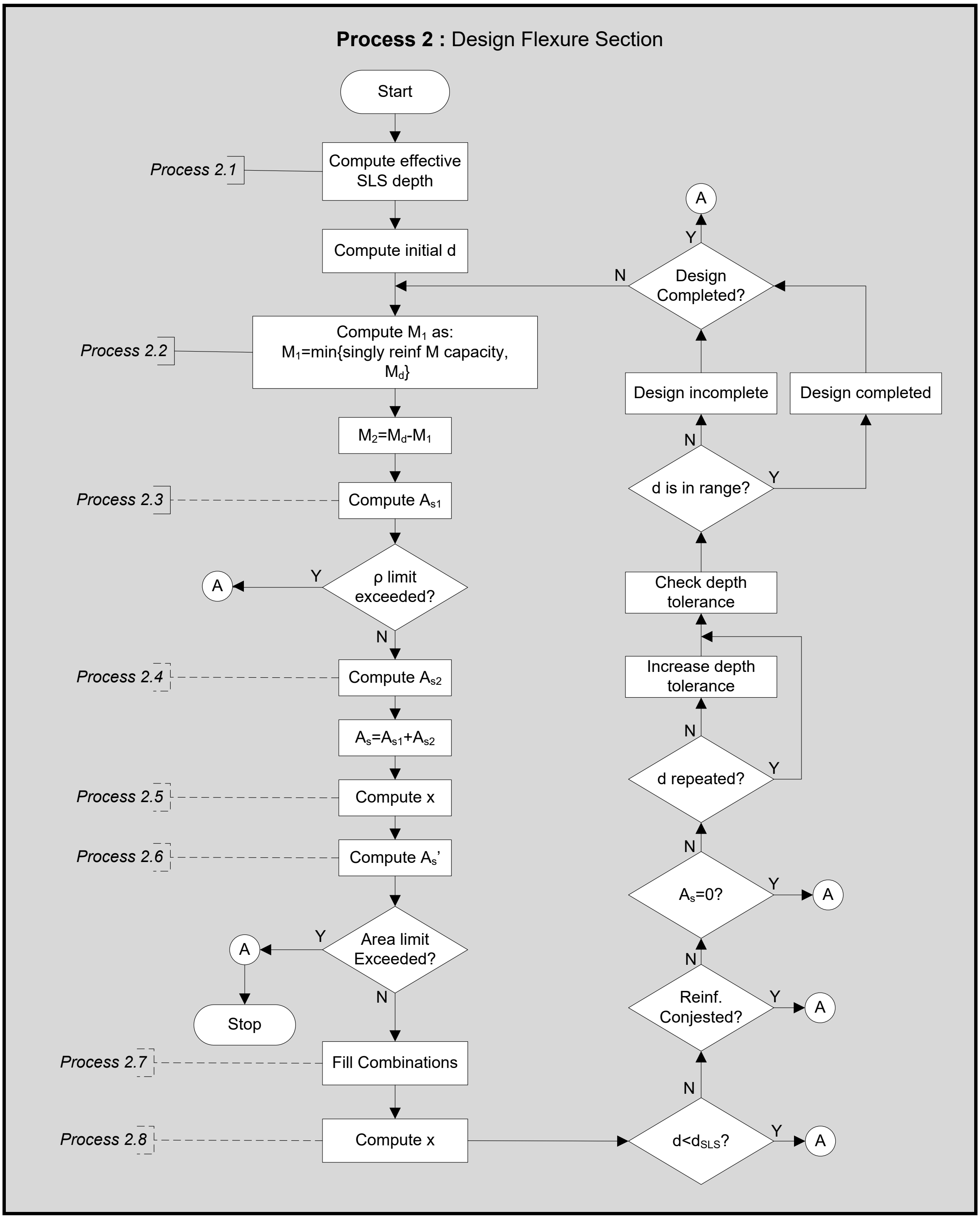
Fig 8-2: Possible Shear sections of common building beams

ESADS is capable of running the design and detailing of any continuous beam analyzed by it. It is not recommended to use ESADS to design continuous beams other than those specified in the scope of the design part of ESADS-Beam, even if it displays a certain detailing.

8.2. Design of Beam for Flexure

After deciding which value of moment diagram we are going to use for a certain interval of length, the next step is to calculate the area of reinforcement to be inserted into the section. In this section we are going to discuss the procedure of ultimate limit state design of flexural members according to EBCS-1995. ESADS uses this method to calculate the reinforcements.

Flow Chart 16: Design of Beam Section for Flexure



8.2.1. Serviceability Limit State

From the serviceability limit state given by EBCS, ESADS deals with only the limits for deflection. The limits for crack are expected to be checked beforehand or afterwards. The scope of ESADS was limited to the deflection only because in this version, we have a primary objective of replacing the former design schemes with simplified, faster ways and not widening the coverage of design.

In EBCS-1995, serviceability limit state for deflection may be ensured either by keeping calculated deflections below the limiting values [Section 5.2.2] or by compliance with the requirements for minimum effective depth [Section 5.2.3]. The calculation of the deflection involves calculation of the effects of temperature, creep and shrinkage.

However, the limit by effective depth only requires calculations based on the span type, effective span length and characteristic strength of the reinforcement steel to be used. Moreover, limiting by using effective depth is most commonly adopted method. Therefore, the effective depth is kept above certain limit to ensure serviceability limit state for deflection.

Requirement of the effective depth is given by EBCS as:

$$d = \left(0.4 + 0.6\frac{f_{yk}}{400}\right)\frac{L_e}{\beta_a} \qquad 8\text{-}1$$

where:

f_{yk} – is the characteristic strength of the reinforcement (MPa).

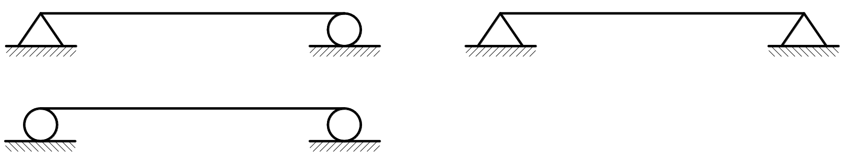
L_e – is the effective span length (As per section 3.7.7 of EBCS – 2 – 1995)

β_a – is the appropriate constant from table 1

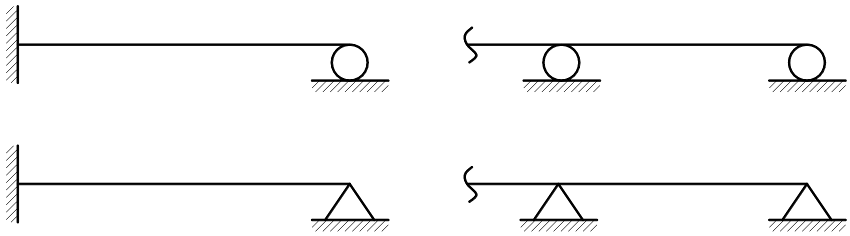
Span Type	Simply Supported	End Spans	Interior Spans	Cantilevers
β_a	20	24	28	10

Most of the time, the gross depth is known in advance. Therefore, the effective depth is only compared to the effective depth calculated from the given gross depth. Calculation of the effective depth is discussed further below. If the calculated effective depth is less than that required by serviceability, the beam is considered as failed.

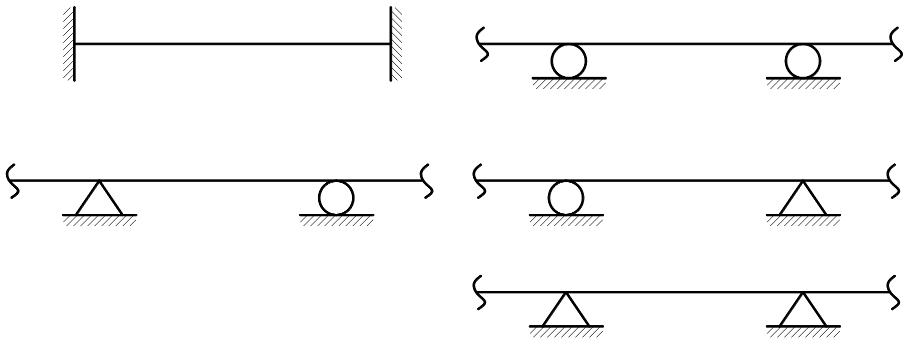
In ESADS, the user is not required to input the span type of each member. The considerations of ESADS to decide the span type of a member is shown below.



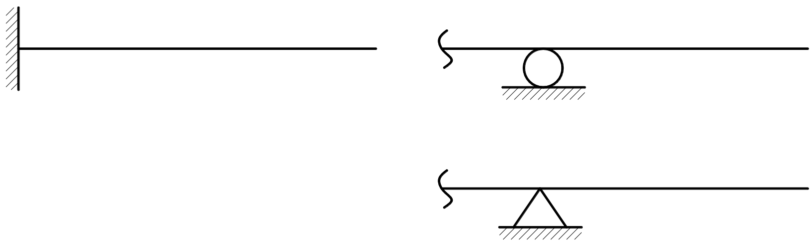
a. Spans considered as simply supported



b. Spans considered as end span



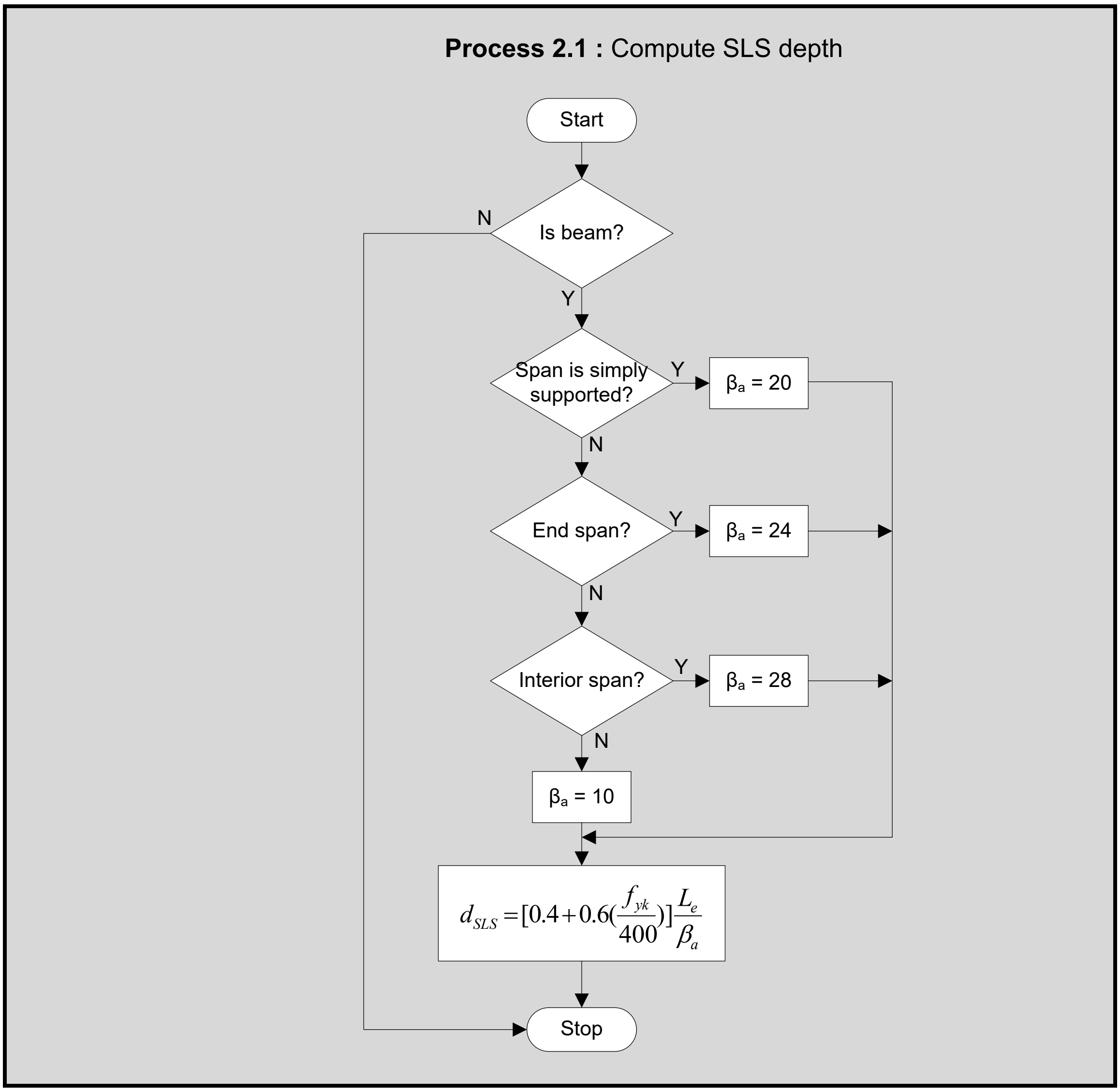
c. Spans considered as interior span



d. Spans considered as cantilever

Fig 8-3: Span types for determination of the serviceability depth

Flow Chart 22: Serviceability Check



8.2.2. Effective Depth

Effective depth of flexural concrete section is the distance from the outer face of compression fiber to the centroid of tensile reinforcement. The centroid of the tensile reinforcement is only known after calculation of the area of steel, which is by itself based on the effective depth. This means there will be some iterative procedure to get the exact area of steel.

As an initial estimate, a single row of the maximum bar size is assumed to calculate the centroid of the reinforcement. Then the effective depth is measured from the center of the row to the top compression fiber.

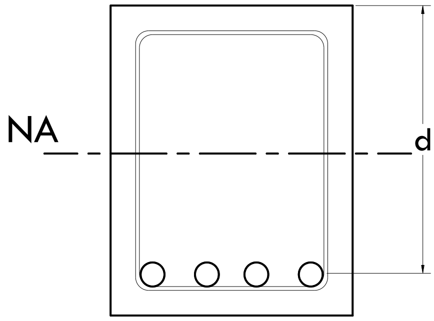


Fig 8-4: Effective depth assumed at the start of design

Having this effective depth, the exact area of steel can be calculated as explained in ultimate limit state topic below. Then the exact number of bars can be arranged so as to get the centroid.

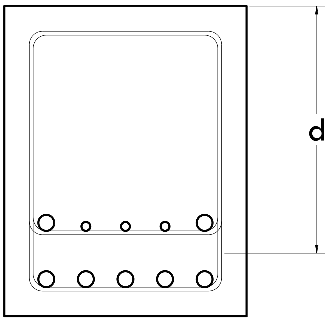


Fig 8-5: Effective depth of a section having more than one layer of bars

8.2.3. Balanced Failure and Code Requirement

As explained in Part I, balanced failure of a concrete section is a condition of section reinforcement whereby the failure of the steel and concrete occurs simultaneously. If the reinforcement amount is added beyond that required for balanced failure, the concrete will be the first to fail before the steel. This kind of failure is relatively sudden as compared to the condition where the steel fails first.

EBCS limits the reinforcement ratio to ensure tensile failure of beam. In its provision moment redistribution of beams [EBCS-2, Section 3.7.9], the limit for reduction coefficient is given for a certain value of percentage redistribution. Taking 0% redistribution, i.e. δ=1.0, the value of x/d may be determined from the formula to be 0.448.

"For continuous beams and for beams in rigid jointed braced frames with span/effective depth ratio not greater than 20."

$$\delta = 0.44 + 1.25\left(\frac{x}{d}\right) \qquad 8\text{-}2$$

The neutral axis depth of a certain section depends on the amount of steel, width of the section and the material strength. For a given width and material, the depth of neutral axis can be varied by varying the amount of steel. The effective depth varies with varying amount of steel.

Therefore, by limiting the value of x/d, EBCS, indirectly limited the amount of steel. From force equilibrium of stress block for purely flexural section;

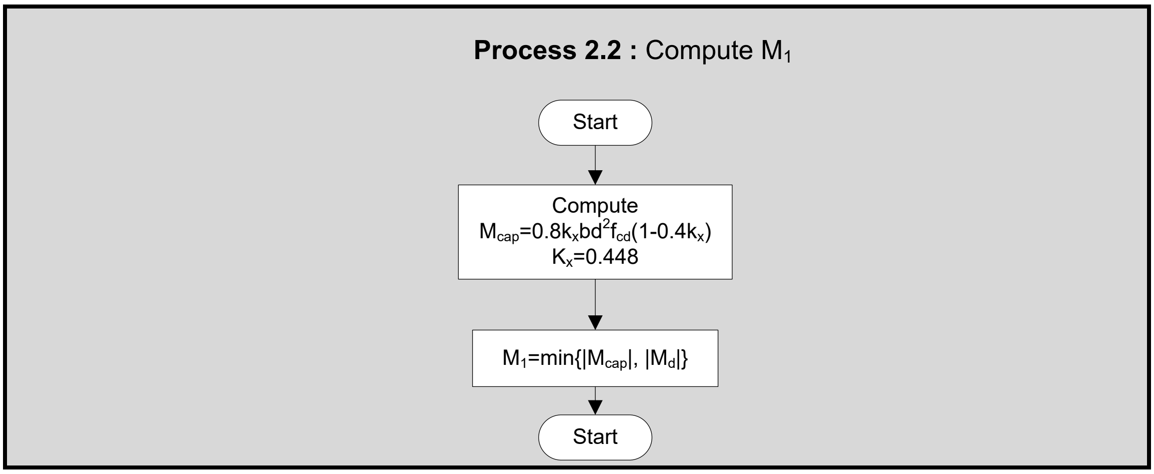
$$x = \frac{A_s f_{yd}}{0.8 b f_{cd}}$$

$$\frac{x}{d} = \frac{A_s f_{yd}}{0.8 b d f_{cd}} \qquad 8\text{-}3$$

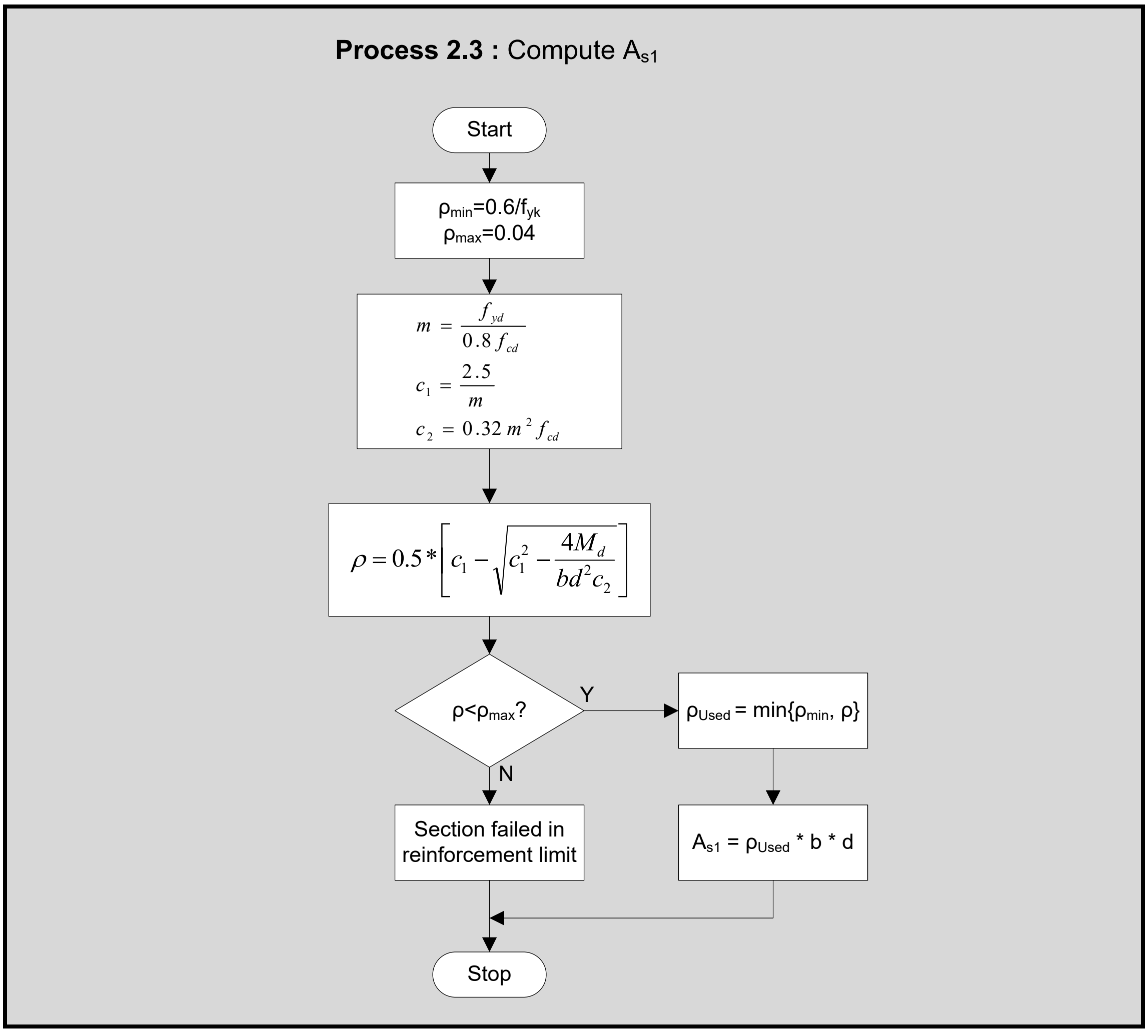
$$\frac{x}{d} < 0.448$$

In the design of flexural sections, the amount of steel satisfying the above requirement may not be sufficient to sustain the applied load. This leads to the use of additional steel both in tension and compression to support the additional moment.

Flow Chart 23: Computing Singly Reinforced Moment Capacity



Flow Chart 24: Computing Area of Steel for Singly Reinforced Capacity



8.2.4. Doubly Reinforced Section

When the design moment of a certain beam section exceeds the singly reinforced capacity, additional tensile steel and another compressive steel is inserted to sustain the extra moment by creating a force couple. The stress level of the compression steel shall be checked against steel yield stress.

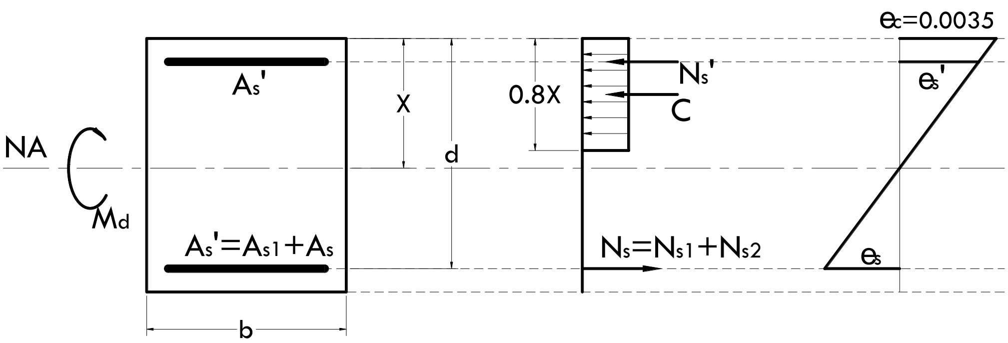


Fig 8-6: Stress strain diagram for doubly reinforced section

$$\sum F_x=0$$

$$N_S' + C = N_S$$

$$A_S'f_s' + 0.8xbf_{cd} = f_sA_S$$

$$x = \frac{f_sA_S - A_S'f_S'}{0.8bf_{cd}} \qquad 8\text{-}4$$

$$\sum M_{N_S} = 0$$

$$M_d = C(d - 0.4x) + N_s'(d - d")$$

$$M_d = 0.8xbf_{cd}(d - 0.4x) + A_S'f_S'(d - d") \qquad 8\text{-}5$$

From strain compatibility:

$$\frac{\varepsilon_c}{x} = \frac{\varepsilon_s}{d - x}$$

$$\varepsilon_s = \frac{\varepsilon_c}{x}(d - x)$$

$$f_s = \varepsilon_s E_s = \frac{\varepsilon_s E_s}{x}(d - x) \leq f_{yd} \qquad 8\text{-}6$$

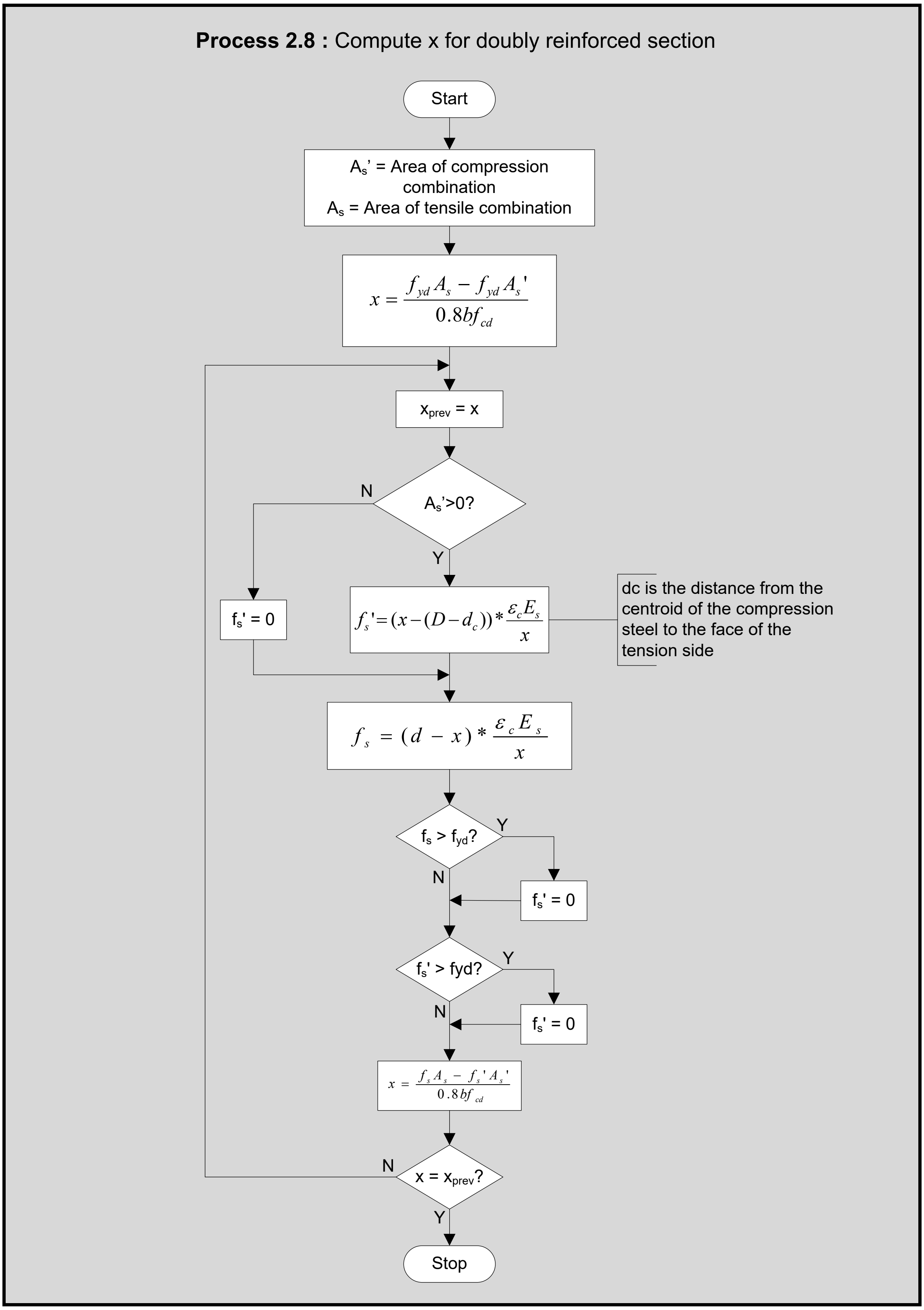
$$\frac{\varepsilon_c}{x} = \frac{\varepsilon_s'}{x - d"}$$

$$\varepsilon_s' = \frac{\varepsilon_c}{x}(x - d")$$

$$f_s' = \varepsilon_s'E_s = \frac{\varepsilon_c E_s}{x}(x - d") \leq f_{yd} \qquad 8\text{-}7$$

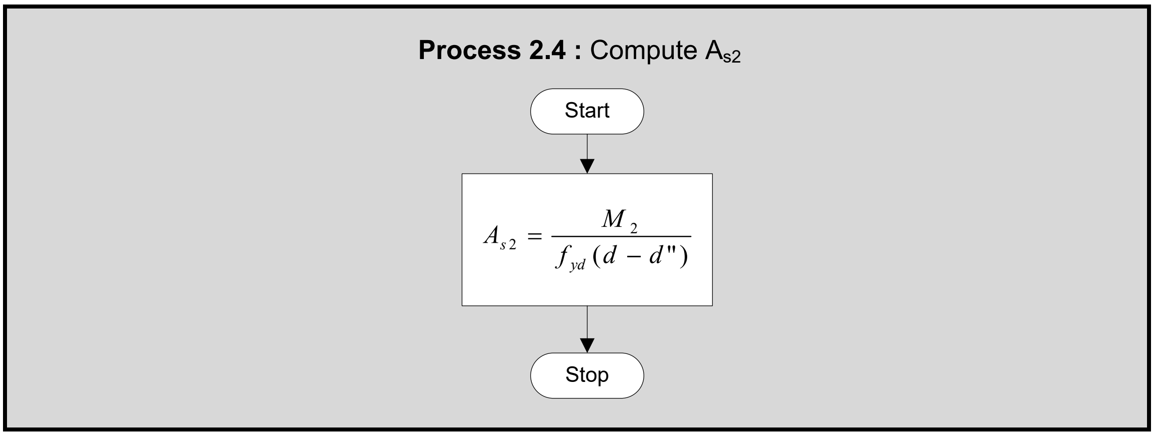
At ultimate load, the tensile steel (eq x.4) has always yielded. However, the compression steel may or may not yield at the ultimate load. If the calculated value of compressive steel stress (eq x.5) exceeds f_{yd}, it is taken as f_{yd} since the maximum capacity of the steel is f_{yd}, even in plastic region.

Flow Chart 29: Computing Neutral Axis Depth

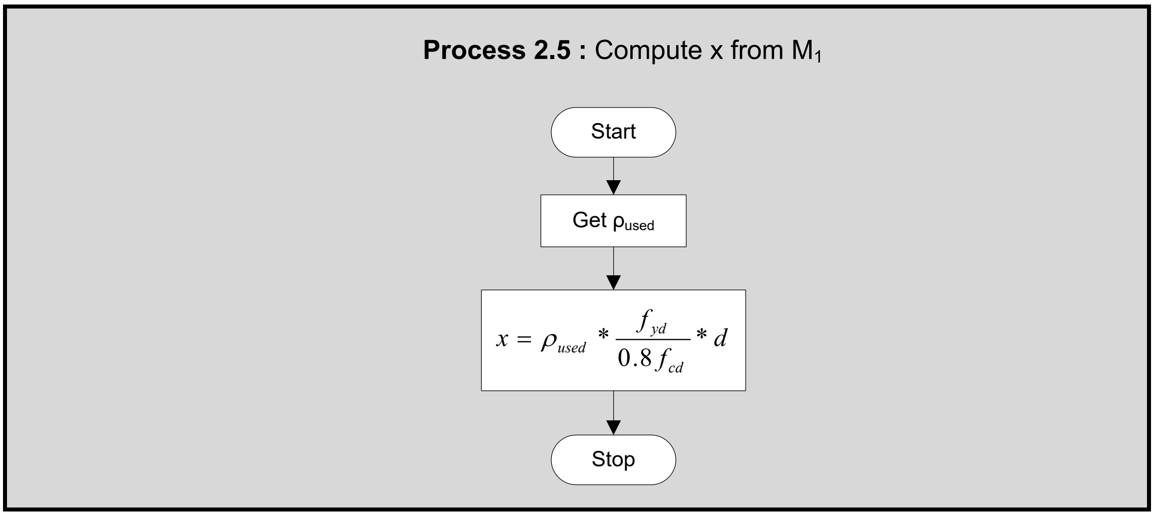


ESADS uses this stress-strain relation for double reinforced section. It first computes the singly reinforced section capacity. Then if this capacity exceeds the design moment, the section is singly reinforced and no secondary reinforcement is required. Otherwise, the extra moment is used to calculate the secondary reinforcement.

Flow Chart 25: Computing Tensile Area of Steel Added as Couple



Flow Chart 26: Computing Neutral Axis Depth from Moment Capacity



For singly reinforced section, eq x.3 reduces to:

$$M_d = 0.8xbf_{cd}(d - 0.4x) \qquad 8\text{-}8$$

Let $\frac{x}{d} = k_x$: Then

$$M_d = 0.8k_x bd^2 f_{cd}(1 - 0.4k_x) \qquad 8\text{-}9$$

The maximum value of k_x, given by EBCS as 0.448, may be substituted to get the singly reinforced section capacity.

$$M_{d,max} = 0.8k_{x,max} bd^2 f_{cd}\left(1 - 0.4k_{x,max}\right) \qquad 8\text{-}10$$

$$k_{x,max} = 0.448$$

Consider the simplified form of equation x.2 for singly reinforced section

$$x = \frac{f_{yd}A_s}{0.8bf_{cd}} \qquad 8\text{-}11$$

Let $\rho = \frac{A_s}{bd}$ and $m = \frac{f_{yd}}{0.8f_{cd}}$

Then equation x.8 becomes

$$x = \rho md \qquad 8\text{-}12$$

But we have assumed $x/d = k_x$

$$k_x = \rho m$$

From equation x.6 and x.9, we get

$$M_d = 0.8\rho m b d^2 f_{cd}(1 - 0.4\rho m) \qquad 8\text{-}13$$

Solving for ρ, we get:

$$\rho = \frac{1}{2}\left[C_1 \pm \sqrt{C_1^2 - \frac{4M_d}{bd^2 C_2}}\right] \qquad 8\text{-}14$$

Where: $C_1 = \frac{2.5}{m}$

$$C_2 = 0.32m^2 f_{cd}$$

EBCS gives maximum and minimum values of ρ, the steel ratio

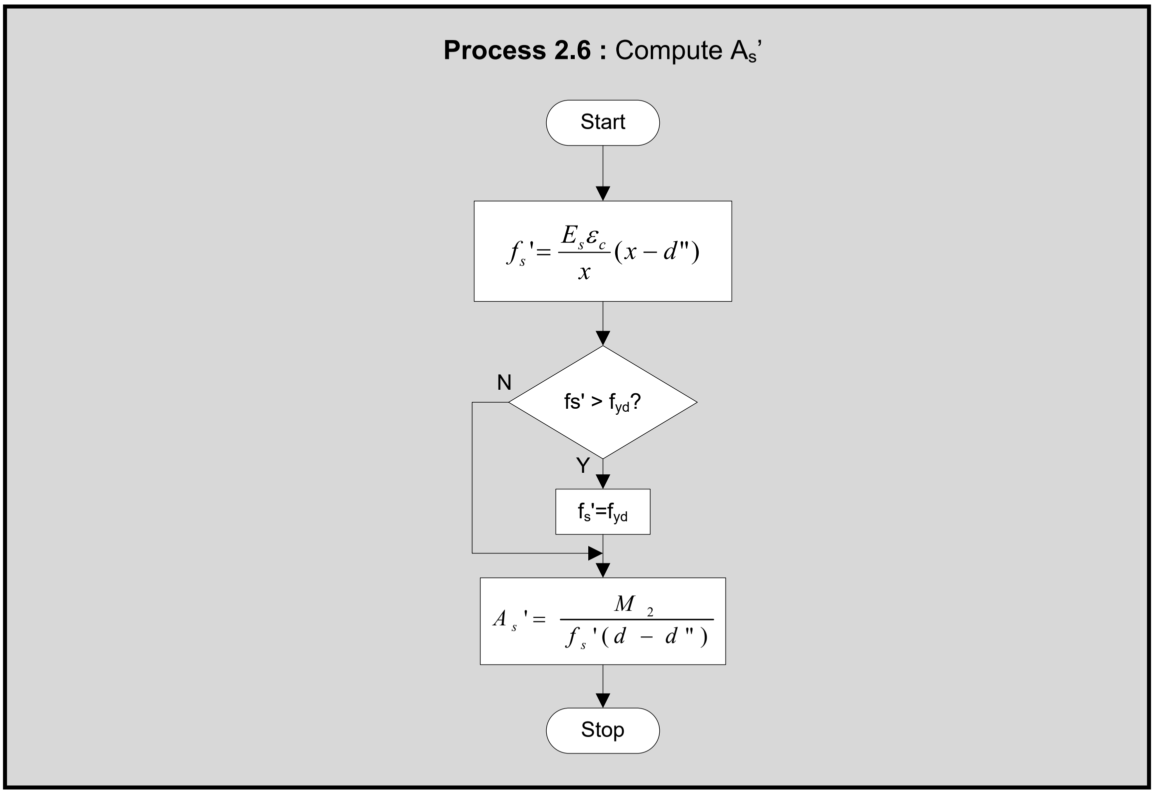
$$\rho_{min} = \frac{0.6}{f_{yd}} \qquad 8\text{-}15$$

$$\rho_{max} = 0.04 \qquad 8\text{-}16$$

Where f_{yk} is in MPa.

ESADS checks these limits after calculating the steel ratio using the above formula. The formula of the steel ration gives two values of ρ. However, the larger value is never chosen. If the other ratio is greater than ρ_{max}, the section has failed. Otherwise the minimum of ρ_{min} and the calculated steel ratio is taken.

Flow Chart 27: Computing Compressive Steel Area



Having the steel ratio, the area of steel can be calculated as:

$$A_{S1} = \rho bd \quad 8\text{-}17$$

If the design moment is greater than $M_{d,max}$, the extra moment must be supported by force couple of the secondary reinforcement.

$$M_d = M_1 + M_2 \quad 8\text{-}18$$

$$M_1 = M_{d,max}$$

$$M_2 = M_d - M_{d,max} \quad 8\text{-}19$$

From fig x.4, taking the equilibrium of the force couple and M_2:

$$M_2 = N_{S2}(d - d) = N_s'(d\text{-}d")$$

$$N_{S2} = A_{s2} f_{yd}$$

$$N_S' = A_S'^{fs'}$$

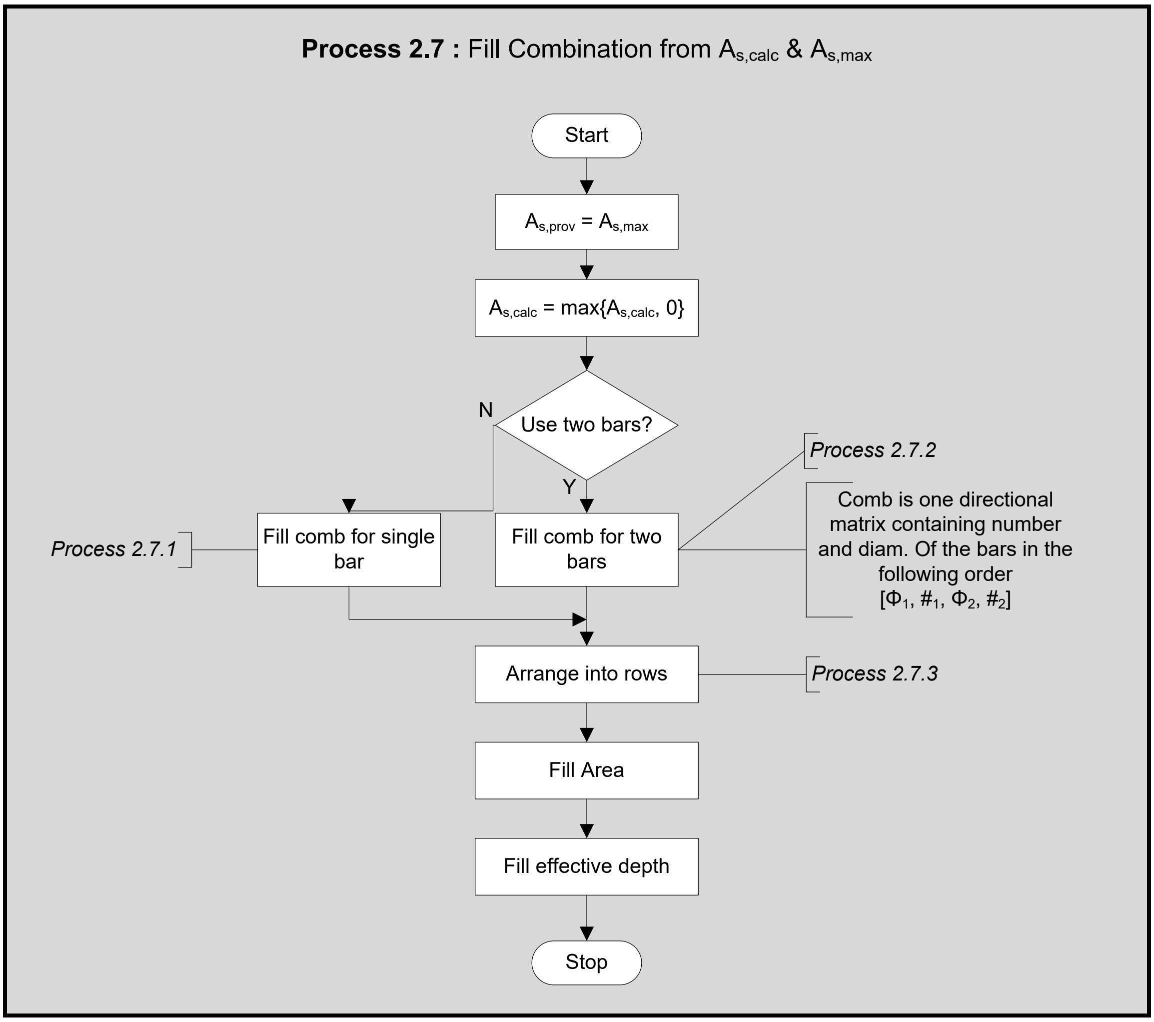
$$\therefore\ A_{S2} = \frac{M_2}{f_{yd}(d-d")} \quad 8\text{-}20$$

Now the compressive steel area may be calculated after obtaining the stress in compression steel from equation x.5.

$$A_S' = \frac{M_2}{f_{S'}(d-d")} \quad 8\text{-}21$$

Now we have the area of both tensile and compression steel. Re-calculation of the neutral axis depth from equation x.2 is necessary to ensure the compatibility of assumed neutral axis depth and that of the actual. With the new value of neutral axis depth, the stress of the compression steel and hence the area of the same is re-calculated. This process is repeated until the variation of the neutral axis depth between consecutive iterations of the process falls below a certain acceptable value.

Flow Chart 28: Filling Combination of Bar



8.2.5. Number and Arrangement of Bars

After the calculation of the area of steel, the next step is computing the most economical bar diameter/s to represent the calculated area of steel. There are different ways of computing the number of bars.

Reinforcements are manufactured in certain predefined diameters. This makes it difficult to get number of bars with area exactly equal to the calculated one. Different diameter of bar may approximate the area better. Combining different bar diameters together approximates even better. Specially, when the calculated area of steel is larger, the possible combination of bars increases. This means it approximates the calculated area of steel.

The table below shows all possible combinations of ϕ14, ϕ16 and ϕ20 for a calculated area of steel equal to 1300mm^2.

$$Available\ bars = \emptyset 14, \emptyset 16\ and\ \emptyset 20$$

Table 8-1: Economical combination of three bar types.

			$A_{s,calc}$	1300	
Diam	**14**	**16**	**20**	**$A_{s,used}$ (mm2)**	**%tage economy**
a_s	**153.938**	**201.0619**	**314.1593**		
max #	**9**	**7**	**5**		
Minimum possible number of Φ14, Φ16 & Φ20 bars in order from left to right	9	0	0	1385.4	93.83
	8	1	0	1432.6	90.75
	8	0	1	1545.7	84.11
	7	2	0	1479.7	87.86
	7	1	1	1592.8	81.62
	7	0	1	1391.7	93.41
	6	2	0	1325.8	98.06
	6	1	1	1438.8	90.35
	6	0	2	1551.9	83.77
	5	3	0	1372.9	94.69
	5	2	1	1486.0	87.48
	5	1	2	1599.1	81.30
	5	0	2	1398.0	92.99
	4	4	0	1420.0	91.55
	4	3	1	1533.1	84.80
	4	2	1	1332.0	97.60
	4	1	2	1445.1	89.96
	4	0	3	1558.2	83.43
	3	5	0	1467.1	88.61
	3	4	1	1580.2	82.27
	3	3	1	1379.2	94.26
	3	2	2	1492.3	87.12
	3	1	3	1605.4	80.98
	3	0	3	1404.3	92.57
	2	**5**	**0**	**1313.2**	**99.00**
	2	4	1	1426.3	91.15
	2	3	2	1539.4	84.45
	2	2	2	1338.3	97.14
	2	1	3	1451.4	89.57
	2	0	4	1564.5	83.09
	1	6	0	1360.3	95.57
	1	5	1	1473.4	88.23
	1	4	2	1586.5	81.94
	1	3	2	1385.4	93.83
	1	2	3	1498.5	86.75
	1	1	4	1611.6	80.66
	1	0	4	1410.6	92.16
	0	7	0	1407.4	92.37
	0	6	1	1520.5	85.50
	0	5	1	1319.5	98.52

	0	4	2	1432.6	90.75
	0	3	3	1545.7	84.11
	0	2	3	1344.6	96.68
	0	1	4	1457.7	89.18
	0	0	5	1570.8	82.76
	Most economical combination			**1313.2**	**99.00**

Therefore, as we can see from the table, the most economical combination of the three bars is 2 Φ14, 5 Φ16 and 0 Φ20 with a percentage economy of 99.00%. The percentage economy of using Φ14 only, Φ16 & Φ20 is 93.83%, 92.37% and 82.76%, respectively. This means, by using more than one bar type we may get a considerable difference in bar economy. It gets even more apparent when used with large area of steel.

This method of economizing reinforcement has a major disadvantage that limits its application. Even if this way of number of bar calculation gives a very optimal economy in terms of the calculated area of steel, using large variations in bar types gets non-realistic results to work with in real construction. Due to this reason, ESADS limits the maximum variation of bars per beam to two. That means the most economical combination of the two bars is used.

Concerning the arrangement of bars into the section, there are some things to be considered. If bars of diameter more than one type are used, it is better to put larger bars near to the outer face of the beam. This results in larger effective depth. Sometimes, bars are arranged in more than one row. In these type of sections, the bars in rows other than the first must be put directly above some bar of the first row. Vertically staggered arrangement of bars results in difficulty of passing vibrator to the bottom of the beam.

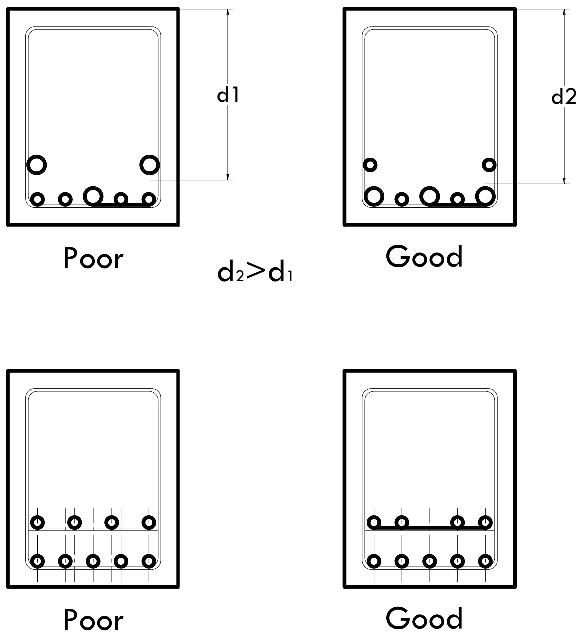


Fig 8-7: Good and poor practices of arrangement of bars in cross-section

After the arrangement of the bars in this way, the effective depth is calculated again. If the calculated effective depth falls in one of the following two cases, the new depth is substituted at the start of the design and the whole section design procedure is repeated.

- The calculated effective depth is less than the assumed one
- The calculated effective depth exceeds the assumed one by a value more than a tolerable margin.

This procedure is repeated until an effective depth that does not fall into either of the above two cases is reached. By the term 'tolerable value', we mean that the extra effective depth resulting in an extra amount of steel that the designer can let go.

Some cases occur whereby the effective depth results in a new effective depth equal an effective depth encountered at some previous iteration. This means a set of effective depth values are being repeated endlessly. To avoid this endless repetition, ESADS uses the safe and most economical depth from those depths in the queue of the repetition.

The economy of the used reinforcement may be expressed as percentage between the calculated area of steel and the provided value.

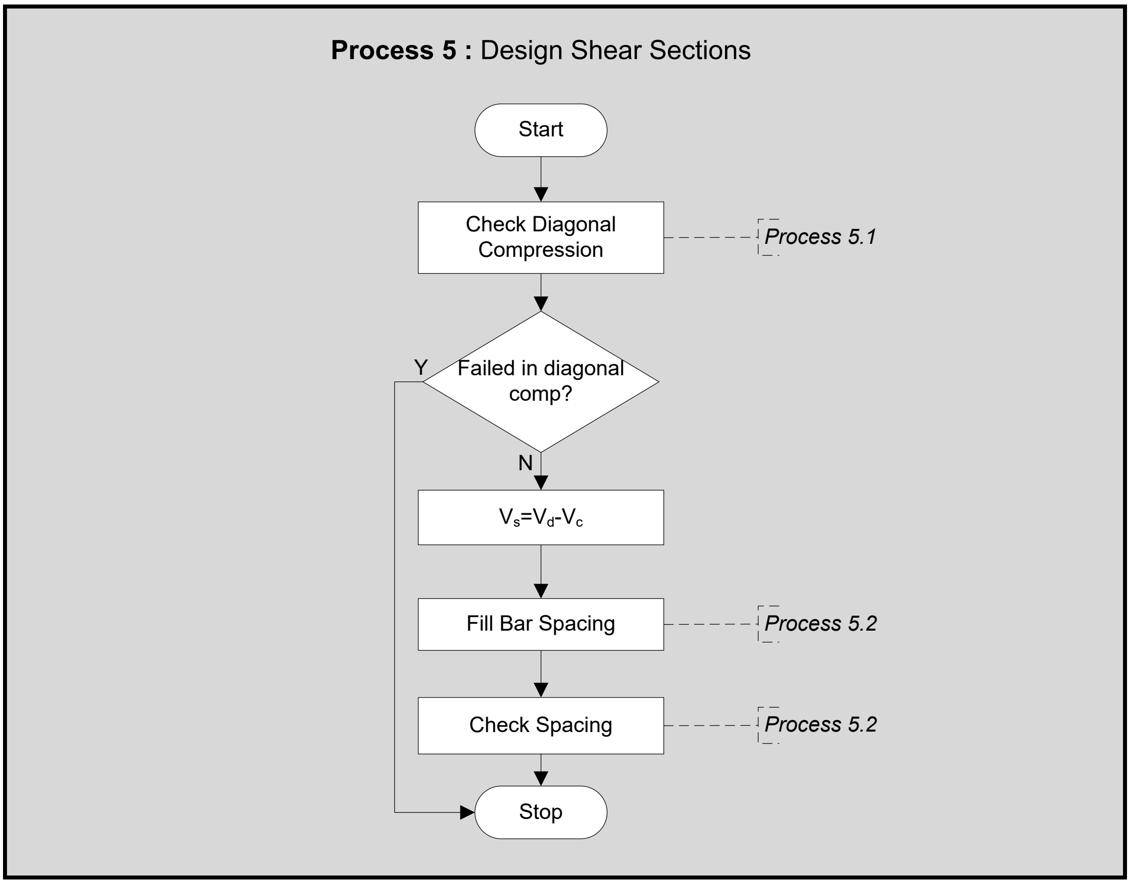
$$\%tage\ economy = \frac{A_{s,calc,tot}}{A_{s,prov,tot}} * 100 \qquad 8\text{-}22$$

ESADS presents the economy of its sections in this way.

8.3. Design of Beam Section for Shear

As explained earlier in this chapter, the shear force value of the SFD is taken starting from a point at d-distance from the face of the support. This is explained here in this section.

Flow Chart 19: Design of each Shear Section



8.3.1. Concrete Shear Capacity

Concrete by itself has some resistance to shear force that depends on the amount of axial load. EBCS-2, 1995 gives the shear capacity of concrete section for sections with or without significant axial load. Within the scope of ESADS, continuous beams are considered not to be exposed to significant axial load.

"The shear force, V_c carried by the concrete in members without significant axial force shall be as:

$$V_c = 0.25 f_{ctd} k_1 k_1 b_w d \qquad \text{8-23}$$

Where:

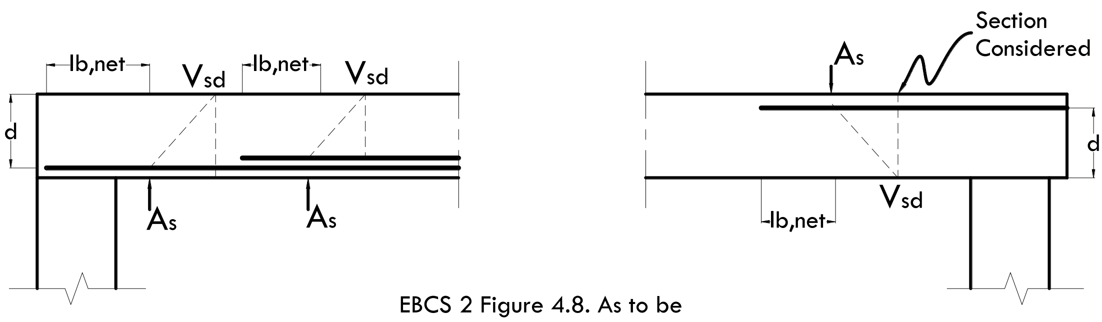
k_1 = $(1+50\rho) \leq 2.0$

k_2 = $1.6 - d \geq 1.0$ (d in meters)

For members where more than 50% of the bottom reinforcement is curtailed, $k_2 = 1$

ρ = $\frac{A_s}{b_w d}$

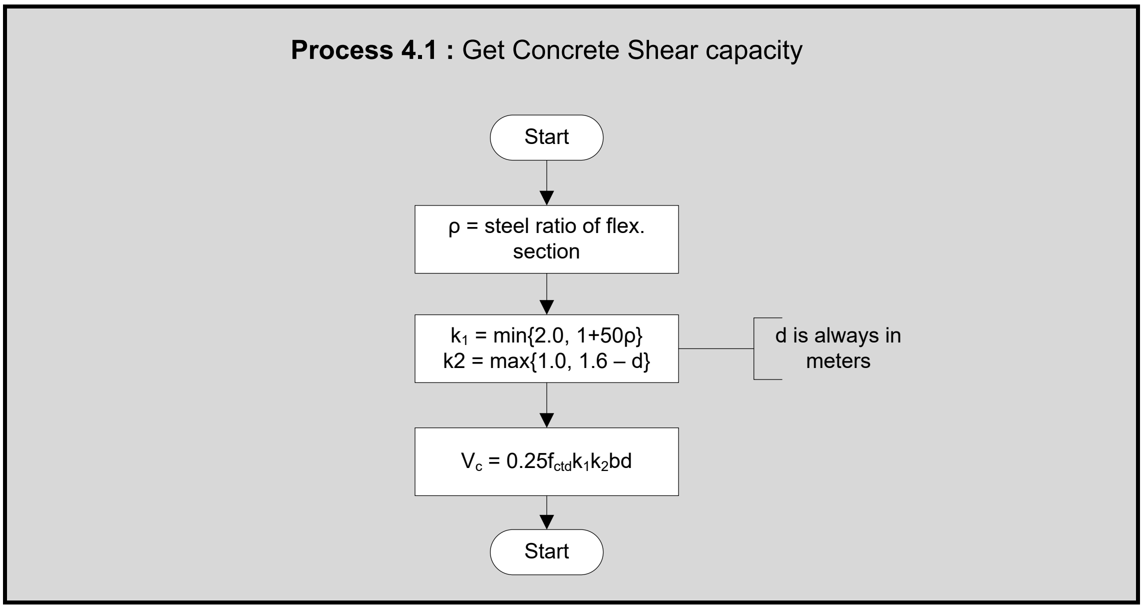
A_s = is the area of tensile reinforcement anchored beyond the intersection of the steel and the line of a possible 45% crack starting from the edge of the section.



EBCS 2 Figure 4.8. As to be introduced in Eq. 4.29

If the applied shear force is greater than the concrete shear capacity stated previously, shear reinforcement may be added to increase the section shear capacity.

Flow Chart 30: Getting Concrete Shear Capacity



However, the concrete must be checked for diagonal compression. EBCS has the following provision for this.

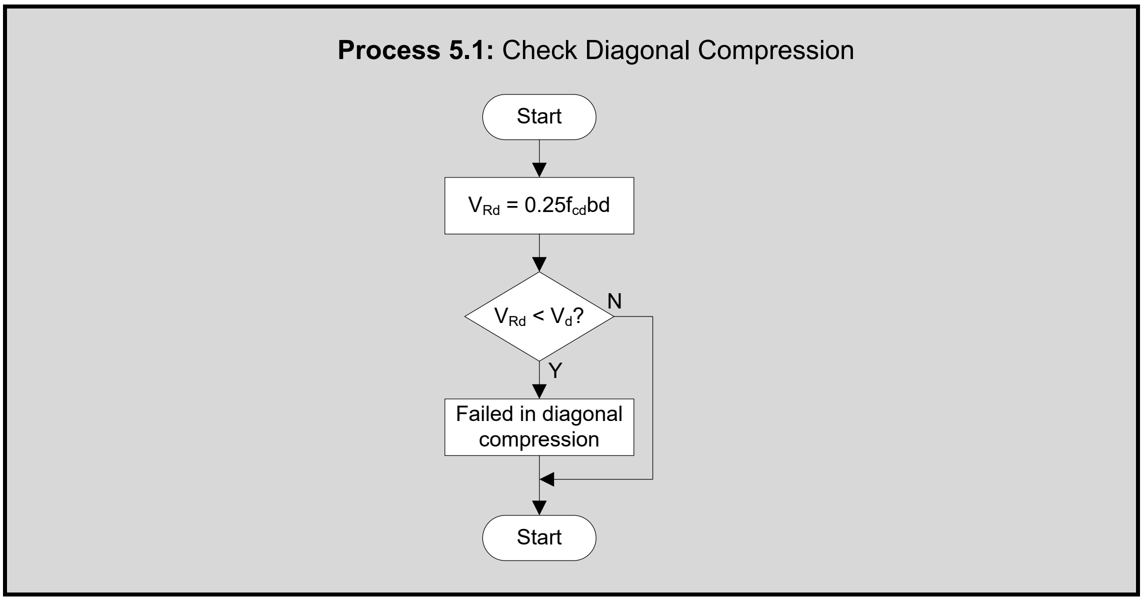
"In order to prevent diagonal compression failure in concrete, the shear resistance V_{Rd} of a section given by Eq 4.28 shall not be less than the applied shear force V_{Sd}."

$$V_{Rd} = 0.25 f_{cd} b_w d \qquad 8\text{-}24$$

Where b_w is the minimum width of the web

If the applied shear is greater than the section's diagonal compression capacity, the shear section is considered to be 'Failed in diagonal compression'.

Flow Chart 31: Checking Diagonal Compression of Concrete



8.3.2. Design of Shear Reinforcement

Within the scope of ESADS, shear reinforcement is provided perpendicular to the longitudinal bars. EBCS provision on this is given below.

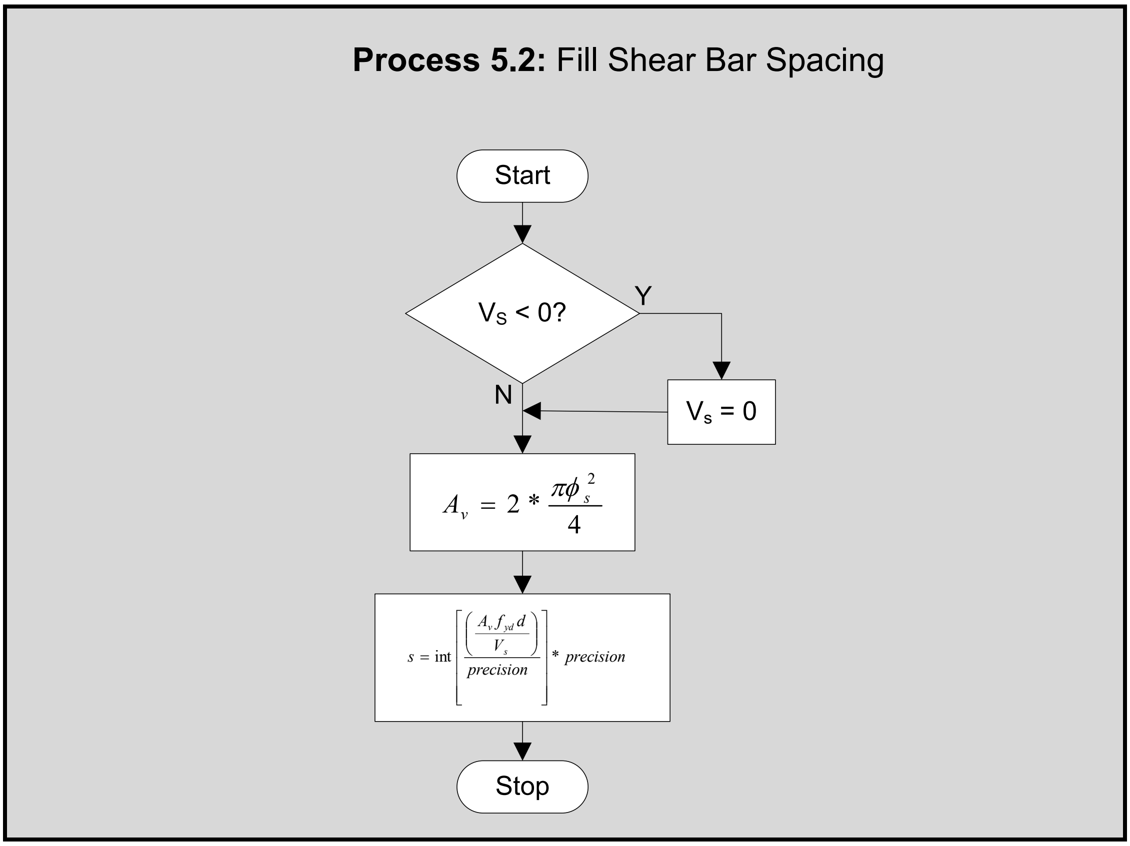
"When shear reinforcement perpendicular to the longitudinal axis is used, its shear resistance, V_S may be calculated as:

$$V_s = \frac{A_v d f_{yd}}{s} \qquad \text{8-25}$$

Where A_v is the area of shear reinforcement within distance S.

The area, Av, is the total area of shear reinforcement, in our case that of the two legs of the stirrup. EBCS gives further provision about the detailing of shear reinforcement, which is discussed further below.

Flow Chart 32: Filling Shear Bar Spacing



8.4. Design Section Optimization

In order to get a workable beam detailing, most designers prefer fewer variation of section details along the beam. Sometimes even a single flexure section and a single shear section may be provided for the entire beam.

Even in the most varied condition, i.e. three flexure section per member, some bars may extend from one section to the adjacent. Span tension bars extend to the end of the member where it may be considered as a compression reinforcement if the support section is doubly reinforced. A relatively rare exception is the case when the amount of compression steel of the support exceeds that of the tensile steel of the span section. In this case, additional bars extend from the support section towards the span for the required anchorage length.

In members with larger load relative to the adjacent spans or in those having large span length, the span moment becomes large relative to the support moments. This may result in doubly reinforced span section whose compression bars extend to the end of the member. These bars may be extended beyond the center of the support for the required anchorage length of the support moment replacing the bars provided at the support.

When we are joining extending bars from section to section, variation of bar diameter may pose significant problem in manipulation. For instance, if the bars extending from the span section are all $\phi 16$ while those at the support are all $\phi 20$, the area of bars shall be compared. If the total area of $\phi 16$ bars that come from the span exceeds that of the $\phi 20$ bars, replace all the $\phi 20$ bars with the $\phi 16$ bars and deduct a number of $\phi 20$ bars, rounded down, equivalent to the $\phi 16$ bars.

Changing the diameter of bars from the designed ones may result in unsafe condition. Especially when area of the steel is varied near to the previous, changing the diameter may also result in significant reduction in effective depth. Therefore, each change must be checked for safety. At the same time the percentage economy changes.

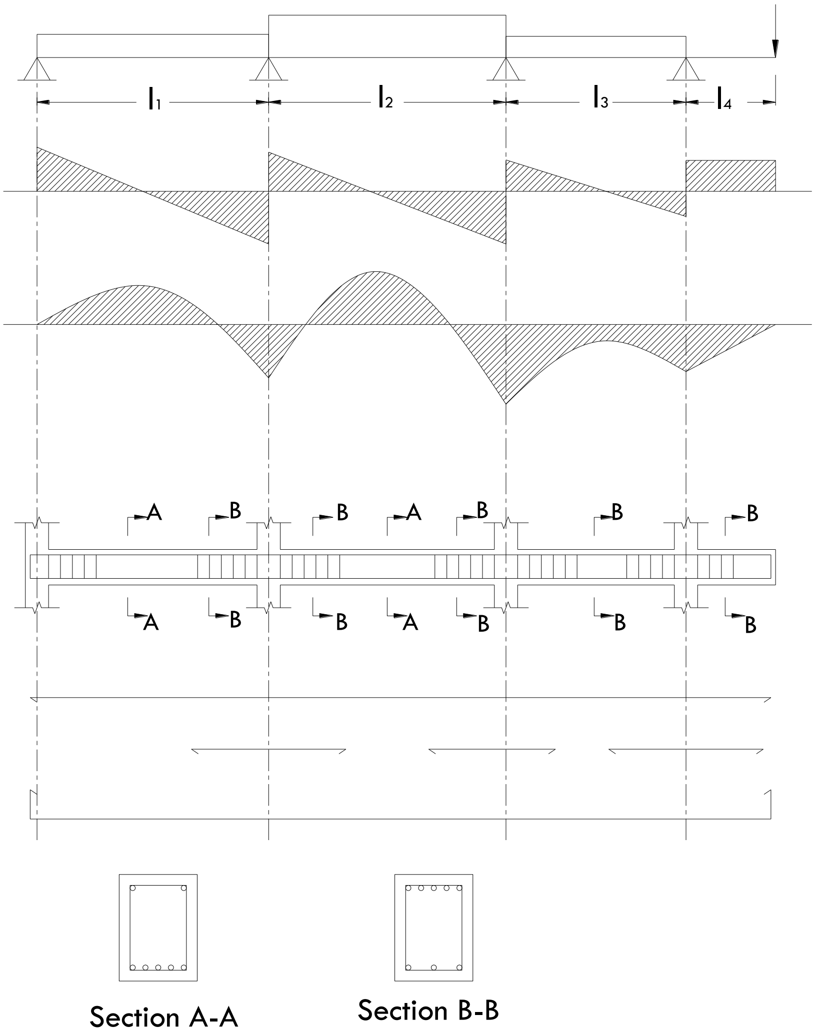
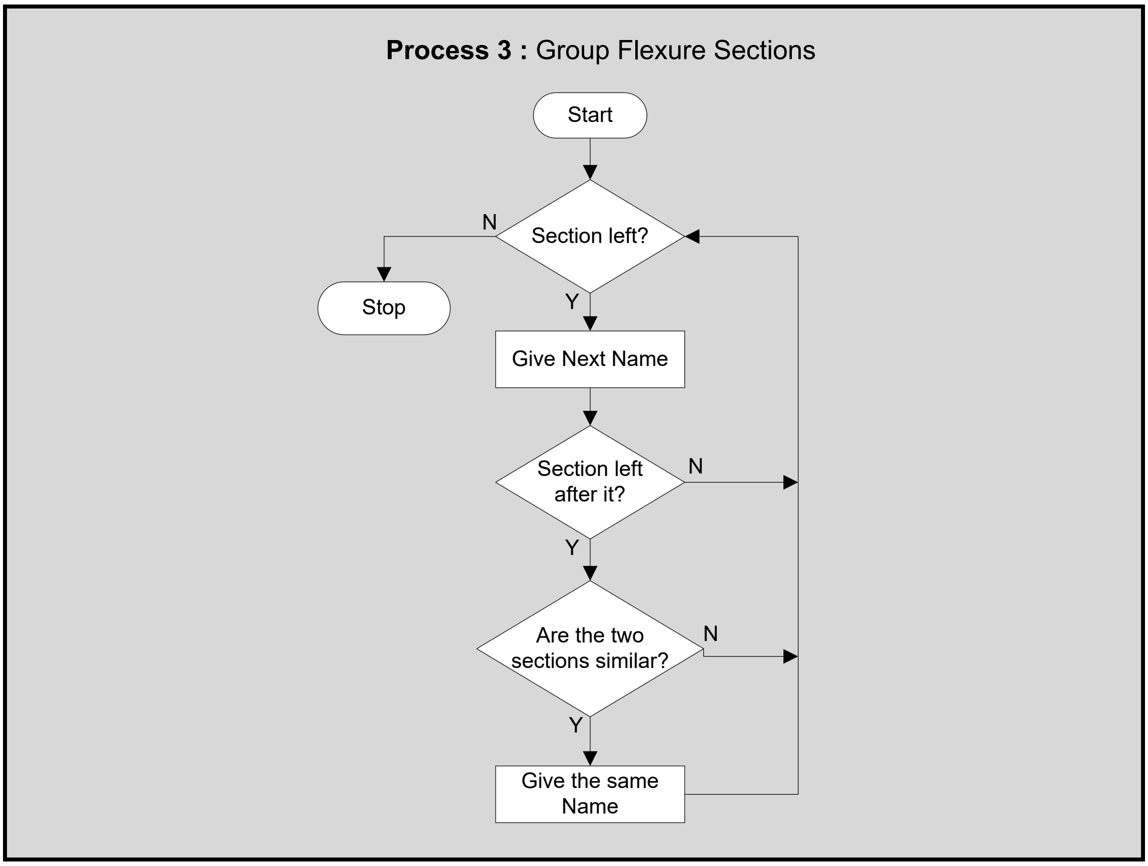


Fig 8-8: Sample bending moment, shear force diagram and detailing

Flow Chart 17: Grouping Sections



When we come to optimization of shear sections, the designer needs to decide the number of shear sections first. Then an exhaustive list of critical shear values of the shear force diagram, i.e. shear values at d distance from the supports of each member and value of V_C. The relative number of required section to the number of critical sections affects the choice of the section intervals and design shear values.

Case I: Number of required section greater than the number of critical shear values.

To increase the number of critical shear values, intermediate shear values shall be taken between the critical shear values until the required number of shear sections is met. A wise way to do this is by inserting a new number between the furthest apart critical points.

For instance, assume the concrete shear capacity is V_C=57kN, and the other shear values are: 170kN, 95kN, 84kN, 210kN, 140kN

Table 8-2: Insertion of critical shear sections

Critical Values	ΔV	Inserted Value	New Critical Values	ΔV	New
57			57		57
	27			27	
84			84		84
	11			11	
95			95		95
				22.5	
	45	117.5	117.5		117.5
				22.5	
140			140		140
	30			30	
170			170		170
	40			40	190
210			210		210

Let us take the required number of shear sections to be eight. Then we have to get two additional shear values between the existing six critical values. Inserting a new value between the furthest apart values would be a wise way. The difference between 95 and 140 is the largest difference. Therefore, a new critical value of 117.5 is now apparent. After the insertion of the new value, the differences are compared again. This time, the difference between 170 and 210 is the largest. Finally, the eighth critical value will be 190.

Case II: Number of required section is less than the number of critical values.

In this case some of the critical values have to be omitted to meet the required number of critical sections. Similar procedure as the previous case may be applied. Only, in this case omit the value with the smaller of two critical values with the minimum difference.

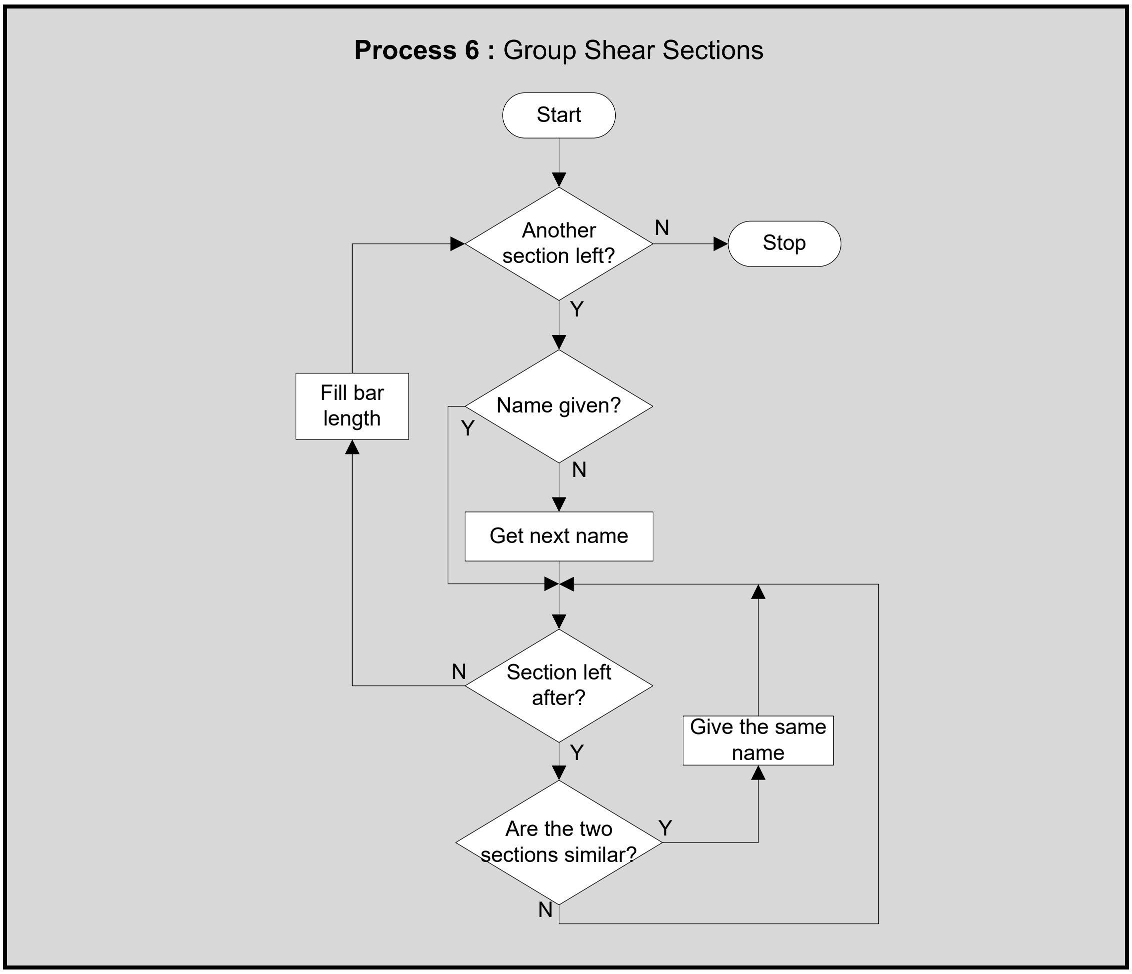
Take the same critical values as the previous but with fewer required number of section, say four.

Table 8-3: Reduction of critical shear sections

Critical Values	ΔV	New Critical Values	ΔV	New
57		57		57
	27			
84			38	
	11			
95		95		95
	45		45	
140		140		
	30		30	
170		170		170
	40		40	
210		210		210

On the first reduction, the critical values nearest to each other are taken. The smaller value is just dropped so that all those sections are going to be included into the next larger section.

Flow Chart 20: Grouping Shear Sections



8.5. Detailing of Continuous Beam

The ultimate goal of RC design is to produce the detail drawing and specifications with which to construct the structure. Beyond the obvious decisions and judgments taken by the designer, EBCS gives some provisions for detailing of continuous beam. Note that all the general provisions provided in Part I still apply for beams.

8.5.1. Longitudinal Reinforcement

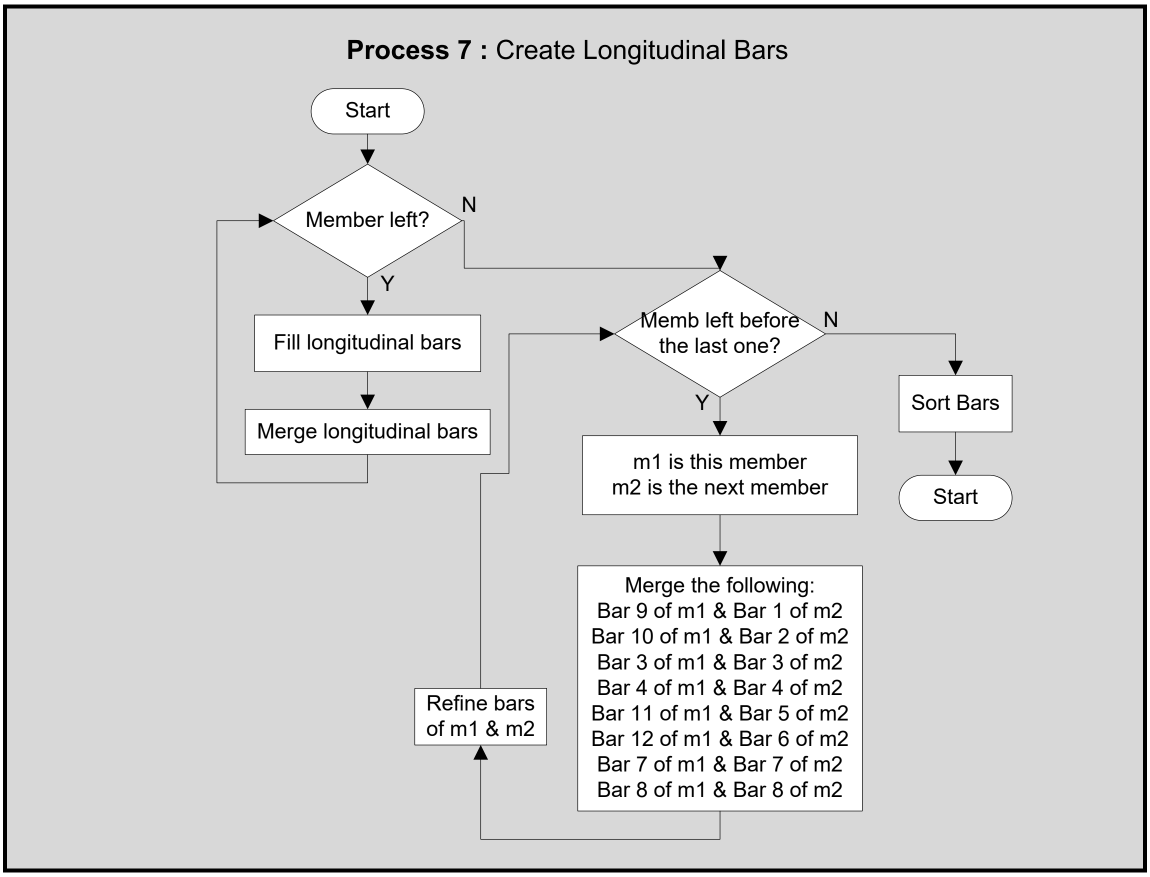
EBCS gives the maximum and minimum values of ρ, the steel ratio:

"The geometrical ratio of reinforcement, ρ at any section of a beam where positive reinforcement is required by analysis shall not be less than that given by: [Section 7.2.1.1(1)]

The maximum reinforcement ratio, ρ_{max} for either tensile or compressive reinforcement shall be 0.04."[Section 7.5.1.1(3)].

Concerning the anchorage length of the longitudinal bars, most of the time a length equal to fraction of the span length is provided and checked for the minimum required anchorage length.

Flow Chart 21: Creating Longitudinal Bars



The relative arrangement between stirrup and the longitudinal bar may be one of two cases. The stirrup may be at the top of the longitudinal bar or vice-versa.

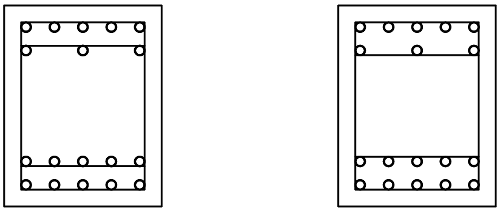


Fig 8-9: Relative Stirrup position

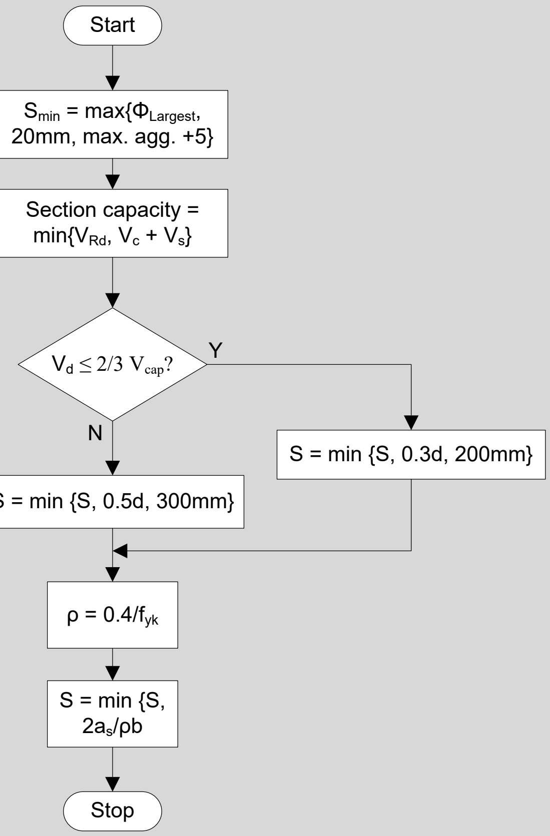
8.5.2. Shear Reinforcement

EBCS gives the following requirement for shear reinforcement.

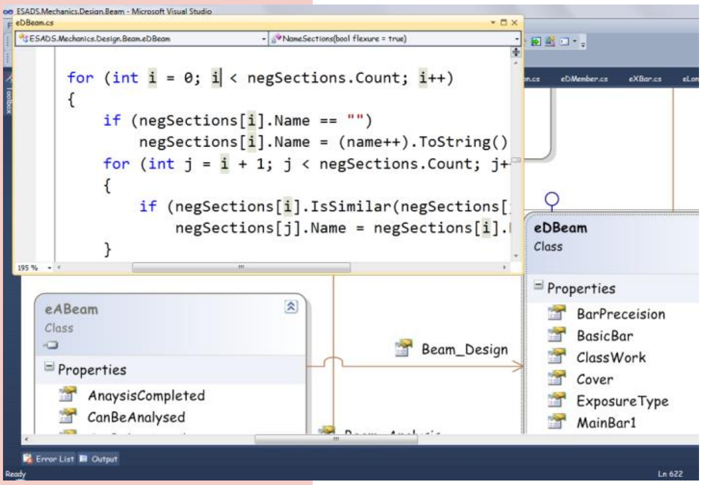
"All beams, except joists of ribbed slabs, shall be provided with at least the minimum web reinforcement given by:

Flow Chart 33: Check Stirrup Bar Spacing

Process 5.3: Check Stirrup bar Spacing



9. Beam – Development



Outline:

- Requirements
- Object Oriented Approach

In this chapter, we will discuss the development process and description of the internal working mechanisms of ESADS – Beam component. In the first section, the algorithm of the program is presented mainly in flowchart form. Then, the user requirements are presented in use cases and activity diagrams. Finally, the alteration of these algorithms and flowcharts into a real software follows in the 'Object Oriented Approach' section.

9.1. Requirements

In this section, we will present the features that have been stated as requirement before the development of the component. This has been classified so as to enable the reader to follow paths.

9.1.1. Structural Analysis of Beam

In this section we will present the features that have been stated as requirement before the development of the component with modifications made through the time of treatment. This has been classified so as to enable the reader to follow the paths.

The requirements are described by the following use cases and activity diagrams.

All the activities that are common with that of other models are discussed in Part I of the document.

Use Case: Starting a new Beam

Primary Actor: User

Goal in Context: To start new beam document

Preconditions: An application of ESADS must be running.

Scenario:

1. The user chooses beam from the new model dialog.
2. The user may change one or more of the following:
   - The number of equal length members to start with
   - The length of these members
   - The support type of each joint
   - The concrete and steel material to be used in design
   - Self-weight consideration
   - Define the default section used in each of the members of the new beam.
3. The system displays define section dialog if the user clicks on 'Define Default Section' dialog.
4. The system considers the newly defined default section for all the members.
5. The system generates new model with the defined member.

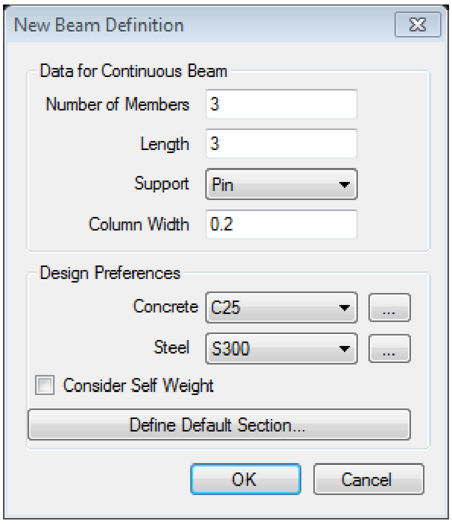


Fig 9-1: New Beam Definition dialog

Use case: Draw Member

Primary Actor: User

Goal in context: Starting drawing member from scratch or continue at the end of existing beams

Precondition: There should be an active beam document

Scenario:

1. The user clicks start drawing member menu item
2. The system draws temporary member horizontally up to the cursor's horizontal position if members exist before.
3. The user clicks anywhere on the drawing to take that point as the start of the whole beam if there is no member before.
4. The user clicks to the right of a previously clicked point or end point of members if they exist.
5. The system adds member between a clicked point and a previously clicked point or end of existing members with default joint and default support width otherwise if ignores it.
6. The system displays a text box if the user presses any numeric key while the temporary member is displayed.

7. The system adds a member with the length value entered in the text box if the user hits enter whole the text box is displayed.
8. The system stops drawing member and hides any temporary member if the user presses 'Esc' or clicks 'stop drawing member' menu item.

Exception:

1. A negative or zero value for member length will not be accepted.

Activity: Drawing Member

Use Case: Editing Member

Primary Actor: User

Goal in context: To change the length of any member

Precondition: There should be an active beam document.

There should be a number of defined members.

Scenario:

1. The user double clicks the member to be edited.
2. The system displays a text box with the current length of the member filled in it.
3. The user changes the value of the text box.
4. The system change the length of the member if the user hits 'enter' while the text box is shown.
5. The system shifts all members to the right of the edited member and the loads on it if the length of the member is changed.
6. The system hides the text box if the user hits either 'Esc' or 'Enter' while the text box is shown.

Exception:

1. If the user enters a zero or negative value for length, the system prevents the changes.

Activity: Editing Member

Use Case: Define/ Assign Member Section

Primary Actor: User

Goal in Context: To define and/or assign a beam cross-section for a number of selected members

Preconditions: There should be an active beam document

A number of members must already be defined

Scenario:

1. The user selects a number of numbers from the previously defied ones.

2. The user clicks 'define beam section' menu item.
3. The system displays the 'define beam section' dialog with the list of previously defined sections.
4. The user chooses to add sew section or modify the selected.
5. The system displays the 'beam section' dialog if the user does action 4 above.
6. The user fills in the name, depth, width, weather to use nominal EI or not and if so, its value.
7. The system displays the area, moment, moment of inertia and the nominal EI, if defined, for the given depth and width of the section when the user wants to see the section properties.
8. The system adds or modifies to the previous list of section if the user accepts the value of 'beam section' dialog.
9. The system saves the list of beam sections if the user chooses either 'Apply', ok or 'assign'.
10. The system assigns the selected section to all selection member is the user chooses to assign.

Exceptions:

1. The absence of name in the beam section dialog prevents the user from accepting the dialog value, seeing preview or section properties.
2. A zero or negative value for depth, width or nominal EI prevents the user from accepting dialog values, seeing preview or seeing the section properties.
3. A duplicate section in the sections list will prevent the user from adding or modifying a section.
4. Attempting to remove a used section will prompt the user and prevents the removal.

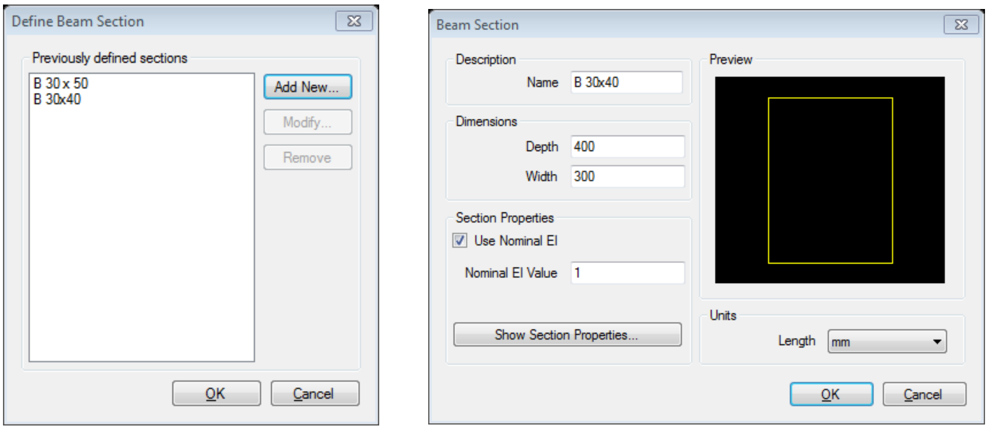


Fig 9-2: Define Beam Section Dialogs

Activity: Defining/Assigning Beam Section

Use Case: Assigning Joint

Primary Actor: User

Goal in Context: To assign a joint type and support width value at selected joints of the beam.

Preconditions: A beam model document must be open and active

Some members and hence some joints must already be defined.

Scenario:

5. The user selects a number of joints
6. The user clicks 'Assign Joint' menu strip item.
7. The system displays "Assign Joint" dialog box.
8. The user chooses joint type and fills in the value of support width.
9. The system changes, correspondingly, all numeric values having units if the user changes either length or force unit.

10. The system assigns the chosen joint type and support width value to the selected joints of the beam if the user accepts the dialog values.

Exceptions:

1. Zero or negative value for the support width will prevent the user from accepting the dialog values.

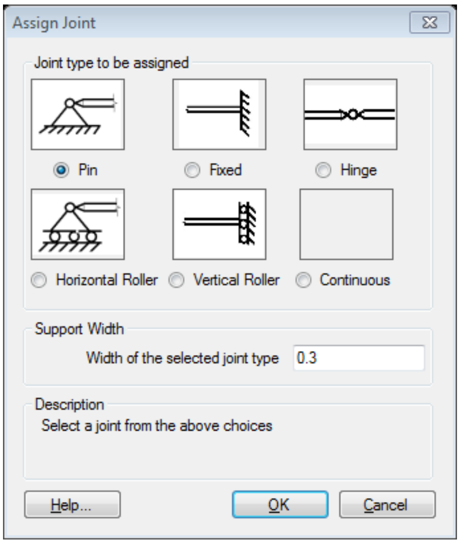


Fig 9-3: Assign Joint Dialog

Use Case: Assigning Member Load

Primary Actor: User

Goal in Context: To assign a load to all selected members

Preconditions: A beam model document must be open and active

A number of members must already be created.

Scenario:

1. The user selects a number of members.
2. The user clicks assign member load menu item.
3. The user chooses the type of load, action type and preference whether it is factored or not
4. The system shows preview.
5. The user chooses between adding, replacing or removing the existing loads on the members.
6. The user chooses the action type and whether to factor it or not.
7. The user chooses the load distance measurement between absolute and relative.
8. The user inputs the magnitude and length parameters of the loads.
9. The system changes, correspondingly, all numeric values having units if the user changes either length or force unit.
10. The system assigns the load to all selected members when the user chooses to assign.

Exceptions:

1. In the case of absolute length, the specified length may lie outside the member. The system moves to the end of the member.
2. Negative length values for any type of load prevents the user from accepting the dialog values
3. Relative distance values less than zero or greater than one will prevent the user from accepting dialog values.

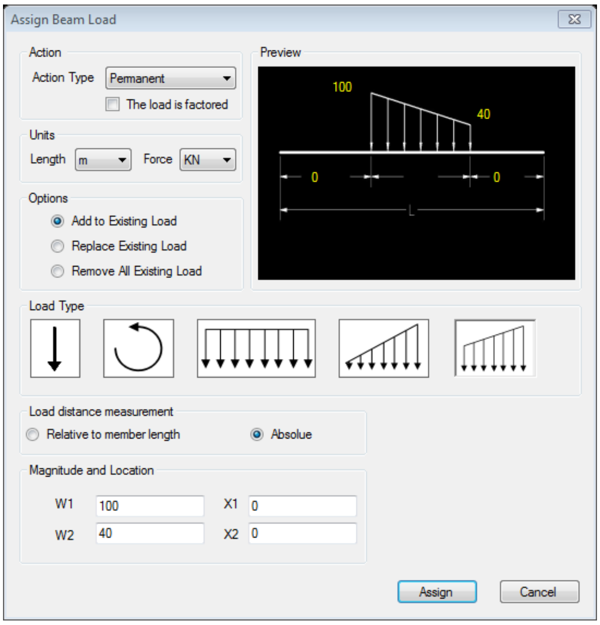


Fig 9-4: Assign member load dialog

Use Case: Assigning Joint Load

Primary Actor: User

Goal in Context: To assign joint load to a number of selected joints.

Preconditions: There should be an active beam document.

There should be a number of defined members and hence joints between them.

Scenario:

1. The user selects a number of joints of the beam.
2. The user clicks 'Assign Joint Load' menu item.
3. The system displays 'Assign Joint Load' dialog.
4. The user chooses between 'Adding the load to pre-existing loads, replacing the existing loads or removing all the loads from the selected joints.
5. The user chooses the action type, load type and whether it is factored or not.
6. The user fills in the magnitude of the load.
7. The system converts the magnitude value accordingly if the user

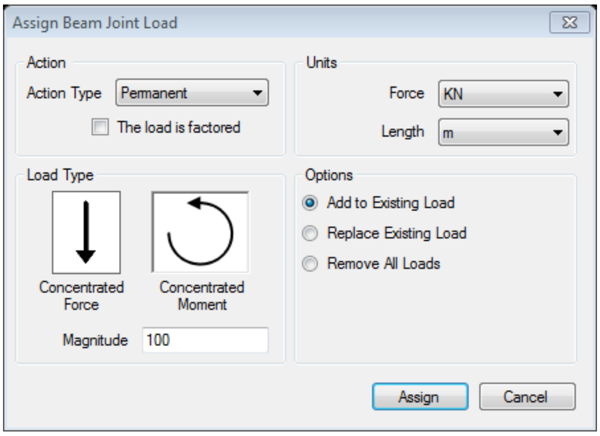


Fig 9-5: Assign Beam Joint Load Dialog

Use Case: Editing Load

Primary Actor: User

Goal in context: Changing the magnitude, direction, location and distribution length of a load over its member.

Precondition: There should be an active beam document

A number of members must be defined

There should be a defined load

The defined load must be visible (load layer must be turned 'On')

Scenario:

1. The User selects the load by clicking anywhere on the load.
2. The system displays the necessary grip boxes when a load is selected.
3. The user clicks on a grip box to turn on and move.
4. The system hides all other loads on the member when any of the grip boxes of a load is turned on.
5. The system regenerates the load drawing as the user moves the grip box.
6. The system accepts the current position of the load if the grip box turned off by click. Returns to original position if it is turned off by hitting 'Escape '.
7. The system displays a text box if the user hits a numeric key while any grip box is on.
8. The system takes the length value, regenerate the drawing and hide the text box.
9. The user double clicks on the text of the load
10. The user displays a text box filled with the magnitude of the load when the user double clicks on the text of the load.
11. The user enters the magnitude of the load and hits 'Enter'
12. The system accepts the magnitude of the load and regenerates the drawing and hides the text box if the user hits enter while magnitude text box is visible.
13. The system rejects the text box value if the user hit 'Escape'.

Exception:

1. A value of length larger than the maximum allowable is automatically changed to the maximum value. The same happens the minimum values.
2. The two extreme grip boxes of a distributed load may overlap which makes it difficult to increase the distribution length.
3. When two or more grip boxes overlap and are turned on together, they all turn off each other and operate in background.

Activity: Editing Load

Use Case: Deleting Drawing Parts

Primary Actor: User

Goal in context: To delete any drawing part from the drawing widow.

Precondition: There should be an active beam document

There should be same deletable drawings on the document.

Scenario:

1. The user selects one or more selectable drawing.
2. The system accepts the selection if the document is not locked.
3. The user clicks 'Delete' menu strip item or hits the 'Del' key on the keyboard.
4. The system deletes all selected items from the drawing if the user clicks 'Delete' or presses 'Del' key and the document is not locked.
5. The system deselects all the selected items if the user hits 'Esc'.

Exception:

1. Attempting to delete a drawing from a locked document notifies the user about the locking.

Activity: Deleting Drawing Parts

Use Case: Selecting/Deselecting

Primary Actor: User

Goal in context: To select an unselected drawing object or to deselect the selected.

Precondition: There should be an active beam document.

There should be same selectable drawings.

Scenario:

1. The user click on the drawing or touches or includes into a selection rectangle.
2. The system ignores the user's action if the document is locked when the user does activity 1 above.
3. The changes the selection state to 'Selected' if the user used selection rectangle otherwise to the opposite of the previous value.
4. The system deselects all the selected items if the user hits 'Esc' from the key board.

Exception:

1. Confusion may be created when overlapping drawings are being managed

Activity: Selection/De-selection of Drawing Parts

9.2. Object Oriented Approach

The classes applied for the beam component are organized. The beam component has its own graphics, analysis and design.

The relationship of the classes is shown in the appendices.

Head Office of Awash International Bank S.C.
Courtesy:
The columns of this medium height building may be design by using ESADS – Column component.

PART III COLUMN SECTION

Chapters:

- Introduction to Column
- Existing Design Practices
- Structural Design - Column
- Software Development – Column
- Checking and Illustration

ESADS V 1.0

10. Introduction to Column

Outlines:

- ❑ Scope
- ❑ Features

In this chapter, we will give a general introduction about column section designing component, so that the reader can have a rough overview of the whole development procedure used and the component itself. Column are vertical compressional load supporting system of the building which support slabs, beams, beams and other supper structural elements. Usually, in addition to compressional load they are also capable of supporting significant amount of bending coming from beams, slabs and even from lateral loads. Therefore, their design procedure involves the combined action of axial and binding load.

In this part, issues related to column component like scope and features, existing design practice, design procedures, software development topics and illustrative design examples using the software are presented in detail.

As columns are expected to support other structural elements, their structural design need to give extreme care for safety, being in economic region. Different codes of practice recommend different structural design and detailing requirements but in the course of this component, EBCS requirements are used as guidelines in the whole development procedure. But, in times when there is lack of detail in EBCS, references are made to related codes like Euro-Code 1992 and ACI.

10.1. Scope

The scope of this component is limited to design and analysis of column section. In other words, this component of the software implements computer program, which can replace the use of design charts provided in EBCS-2 Part-2. When dealing specifically, we can classify the scope as scope related to design loads considered, geometry and detailing type. We will describe each of these specific scopes in detail in the following subsequent topics.

10.1.1. Scope Related to Design Loads Considered

The software designs column section for axial load combined with both uniaxial and biaxial bending. There will be no consideration to shear and torsional moment. Therefore, the user must design the section for shear and torsion separately. But, only for the sack of detailing the user is required to input the shear reinforcement diameter.

10.1.2. Scope Related to Geometry

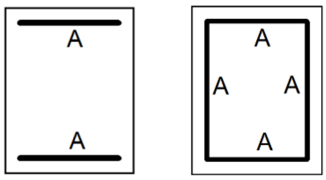
Related to geometry of the column section, the software is only developed for rectangular columns sections reinforced in different detailing pattern.

10.1.3. Scope Related to Detailing Type

This software is capable of designing rectangular column section with different detailing types. The detailing types, which can be designed using this software, are listed below. Note that, identification for each detailing type is given as Type1, Typ2, Type3 and Type4 and the same identification will be used to describe these detailing types in the entire document. Generally, under detailing the scope is limited to:

- ✓ Symmetrically reinforced arrangement.
- ✓ Discrete reinforcement units.
- ✓ Uniform horizontally and vertically distributed reinforcement arrangement.

A. Uniformly Distributed Reinforcement



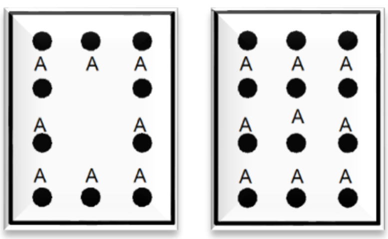
Type-1 Type-2

Figure 10-1. Different type of uniformly distributed reinforcement

In Type1 the reinforcement are equal in opposite top and bottom side of the column. In Type2 the reinforcements are uniformly distributed over the all faces of the column and the same reinforcement area is used in each face.

B. Discrete Reinforcements

The following two pictures shows symmetrically arranged point reinforcements detailing types, to which this software is capable of.



Type 3 Type 4

Figure 10-2.Different type of discrete reinforcement arrangement

In the first detailing type, Type3, the user is only allowed to input number of bars in horizontal and vertical direction. Similarly, in the second (Type4) the user can input number of rows and columns of bars. Then, the bars are uniformly distributed in the periphery and in the entire cross section respectively, assuming cove.

10.2. Features

Features of the software listed here are those features, which are expected to be unique to ESADS Column Design Component as compared to other softwares, manual methods, and design aids, commonly used to design column section before.

The features are described under the following categories.

- ✓ Detailing
- ✓ Interaction Diagram and Surface.
- ✓ Intermediate Outputs
- ✓ Exporting to AutoCAD2007
- ✓ Others

10.2.1. Detailing

Previously, under scopes we have tried to explain different detailing types, which are integrated in ESADS column design component. Now features provided in each detailing type will be discussed, so that the user may take the advantage of using them. Depending on the requirement of the designer, the following detailing features are provided with regard to the detailing type selected.

- ✓ Calculate the most economical bar diameter among list of bars provided by iteration.
- ✓ Display calculated and provided area of steel with percentage of economy achieved
- ✓ Checking maximum and minimum reinforcement requirement before generating the final detailed drawing.
- ✓ Presenting the detailed drawings pictorially so that the user can have a better visualization of the design output.

Detailing procedure as a Feature:

When detailing, initially bar diameter is taken among the list of bars preferred by the user and the design is carried by computing cover parameters in both dimensions of the column i.e. h' and b'. But, after calculating the area of reinforcement we may find that initially assumed bar diameter will not be economical or safe. Therefore, we should have to choose other diameter which can economically and safely fit the calculated area of steel. While doing so, since the bar diameter is changed, we should calculate the new cover parameters (h' and b') and revise the design. To solve this routing iteration the software integrated a functionality, which will automatically iterate the design until the calculated bar diameter and assumed bar diameter are equal. For further detail, refer Section 5.7

10.2.2. Interaction Diagram and Surface

Interaction diagram are two dimensional curves, which represent the flexural capacity of the uniaxial column at different axial load. Analogously, Interaction Surfaces are 3D surfaces, which express the relation between axial load and the two biaxial moments.

A. Interaction Diagram

ESADS draws the interaction diagram for particular type of biaxial column under design to show the user some critical point on the diagram like,

- ✓ Point of maximum axial capacity
- ✓ Point of balanced failure
- ✓ Point of pure flexural capacity
- ✓ Point at which the design axial load and the design axial moment meet on the diagram to check safety and the failure, compression or tension failure.

Interaction diagram is draw for both calculate area of steel and provided area of steel. This will also let the user to see the overall economy pictorially and the available extra strength.

In addition the user is able to calculate the flexural capacity of the cross section at a given axial load using both the calculated or provided area of steel as he preferred

B. Interaction Surface

ESAD only integrated two dimensional presentation of interaction surface by taking the wireframes of the surface at different M_x/M_y ratio. Since our graphics is not develop to such extent that it can sow the 3D surfaces, we conventionally presented 3D surface of Interaction Surface by drawing a curve connecting points of constant M_x/M_y ration or points if constant inclination. The curve connecting these points of constant inclination is called Isoclines. Similarly, the curve connection point of constant axial load is called Load Contour. See the following figure for Isoclines and load contour in a typical Interaction Surface.

In addition ESADs column design component also let the user to get magnitude of load carried by the section given some of parameters. For example, the user provides M_x/M_y ration and Axial load, then the program will display M_x and M_y in the interaction surface.

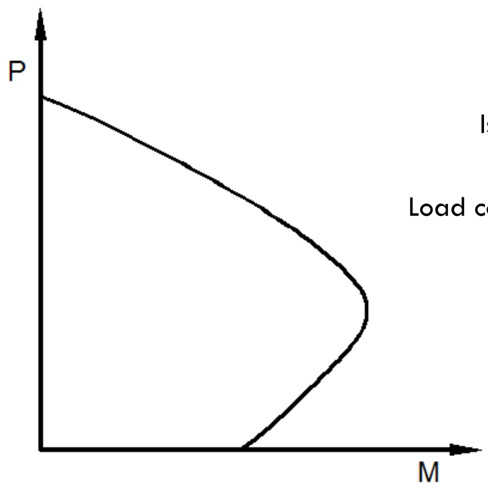


Figure 10-3 Interaction Diagram

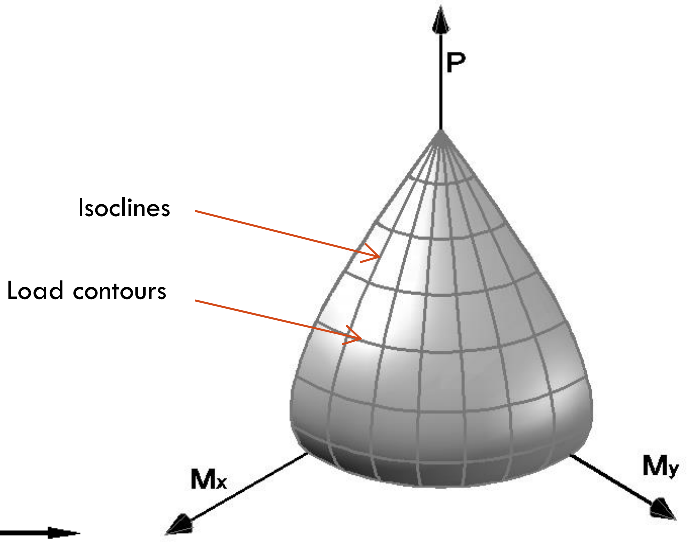


Figure 10-4. Integration Surface

10.2.3. Intermediate Values

The following intermediate value of design cab be displayed for user for checking and sometime academic purpose.

- ✓ Depth of neutral axis required at a given moment and axial load in uniaxial columns.
- ✓ Neutral axis depth measured perpendicular from top most compression fiber to the inclined neutral axis and angle of neutral axis.

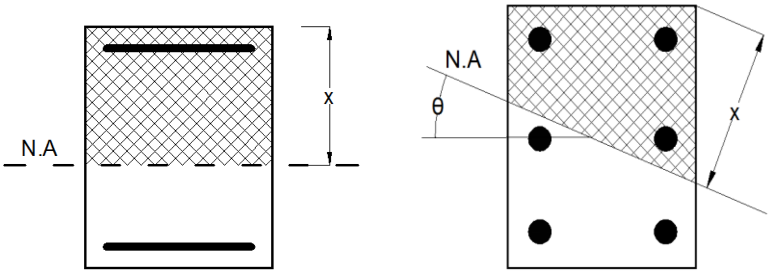


Figure 10-5 Intermediate outputs of uniaxial and biaxial column

10.2.4. Exporting the Detail to AutoCAD2007

In actual design the whole design procedure should end by the preparing detailed structural drawings. Depending on the preference of the designer the software can export detailed drawing of the column section to AutoCAD2007.

11. Existing Design Practices

Outlines:

- ❑ Design Charts

In our country, Ethiopia, most designers design reinforced column section using design aids provided in EBCS-2 Part-2, while other directly use outputs from design softwares like SAP and ETABS. But, these softwares are not developed based on EBCS; rather they use other design codes like EUROCODE 2 -1992, from which our code gets it origin.

In column section designing usually Design Aids or some computer program is needed. This is because of the complexity of the design procedure involving two or more unknowns, which are related nonlinearly.

11.1. Design Charts

Considering column design charts provided in EBCS-2 Part-2 we can list several limitations. It is evident that the charts greatly simplify the design of column section. But, they inherently face the following main limitations. These limitations are the core of our need to develop this column design component.

Generally, the limitations seen in the design charts can be categorized into two depending on their design scope and the way of using them.

1. Limitation directly related to the charts.
2. Limitation related to the end user.

11.1.1. Limitation Related to Charts

Here we will list those limitations of the design charts, which are directly related with their scope and way of presentation.

C. Discreteness

Discreteness can be seen in term of

- ✓ Steel grad
- ✓ Relative cover ratio
- ✓ Mechanical reinforcement ratio

Steel Grad:

For example, uniaxial charts from 1 to 15 in EBCS-2 Part-2 are developed for steel grade S460 refer EBCS-2 Part-2 Page 56.

Relative Cover Rations:

All the design chars are develop for limited relative cover ratios. These are $\frac{h'}{h}$ = 0.05, 0.01, 0.015, 0.02 and 0.25.

Mechanical Reinforcement Ratio (ω)

The design charts are prepared for limited ***ω values raging from ω = 0 up to ω = 1.2. When we get intermediate value we do interpolation or take the larger value for safety.***

D. Specific to Detailing Requirements

We can design column only using the limited types of reinforcement arrangement described in the chart. For example, we cannot design column section using charts having the detailing type Type4 described under Section 2.1.3 (B).

When the axial load is dominant relative to the moment the following detailing arrangement will eliminate the reinforcement congestion if all bars are going to be distributed around the periphery.

E. Conservativeness

If we check the outputs from the charts manually we can clearly see that they are somewhat conservative.

F. Geometric Limitation

The design charts are only prepared for rectangular or circular column section. Though we have not eliminated this limitation in our program also, here we described it only for the sack of completeness.

11.1.2. Limitations Related to User

These limitations of the charts related with the user as a result of the poor way of presentation in the charts. For example the user is highly exposed to the following mistakes.

- ✓ Missing the correct point in the chart.
- ✓ Following wrong curve in the chart.
- ✓ Missing the correct chart and following other.
- ✓ Other human imperfections.

Some of this limitation may be avoid if the charts are presented in a tabular format. As much as possible, our software will eliminate most limitations seen in the charts.

12. Structural Design – Column

Outline

- Introduction
- Reinforced Column Sections in General
- Methodology
- Design Procedure for Uniaxial Column
- Design Procedure for Biaxial Column

In this portion, general design procedure used to design both uniaxial and biaxial column is discussed. First, mechanics of reinforced concrete design for column is described with some more code requirements. Then, the methodology that we have used for ease of implementation for computer is presented. Finally, design procedures for both biaxial and uniaxial column are given on stepwise basis. In the course of this chapter, assumptions made in each step are noted and possible reason is also given.

12.1. Reinforced Column Sections in General

As we have tried to describe in the introductory part columns are vertical structural member subjected to both axial compressive load with or without flexure. Depending on the way of loading they are classified as uniaxial or biaxial. Uniaxial columns experience axial load and bending only in one axis. But, biaxial columns support axial load and bending in two principal axes.

The principle of stress and strain compatibility used in the design of reinforced concrete beam is also applicable in columns. In design of beams ductile failure is necessity, but; in reinforced columns the axial compression may dominate and the failure may be a compression failure.

Based on EBCS-2 the ultimate limit state of reinforced column design is subjected to the following assumptions.

1. Linear stress strain distribution is exists across the depth of the column.
2. Reinforcements are subjected to the same variation of stain as adjacent concrete.
3. The tensile strength of the concrete is neglected
4. The behavior of the concrete under compression is shown by tress strain diagram of Figure 12-1
5. The stress stain relationship of reinforcements is given by the Figure 12-2
6. The strain diagram at ultimate limit stat is given by Figure 12-3

a) The maximum compressive strain in the concrete is taken to be 0.0035 for bending (simple or compound) and 0.002 in pure axial compression

b) The maximum tensile strain in the reinforcement is taken to be 0.01

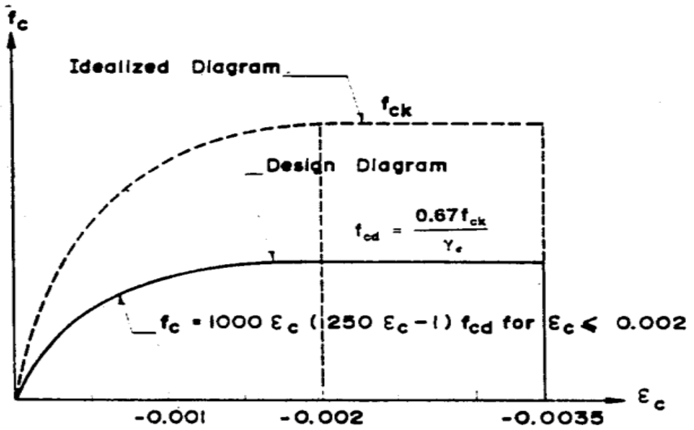


Figure 12-4 Stress strain diagram of concrete

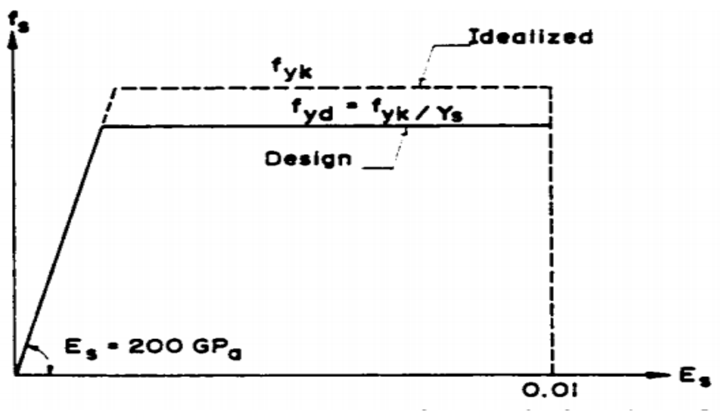


Figure 12-5 Stress strain diagram of steel

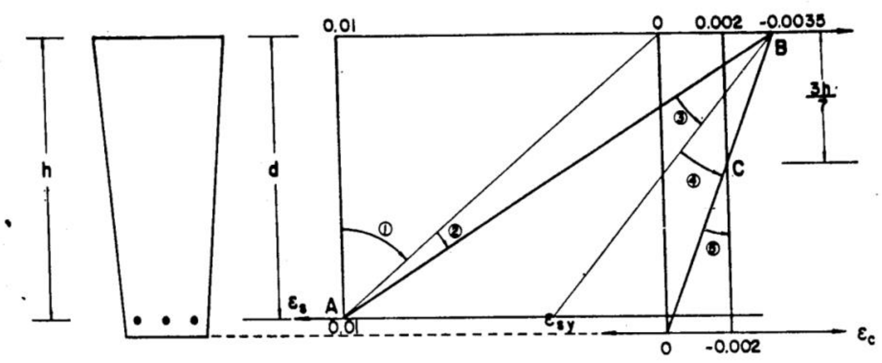


Figure 12-6 Strain diagram at ultimate limit state.

For the sack of design simplicity, usually rectangular stress block is used to present the effect of parabolic stress distribution. In EBCS-2 the following figure is given for rectangular stress block. In the course of this component a rectangular stress block is used to calculate the force taken by the steel.

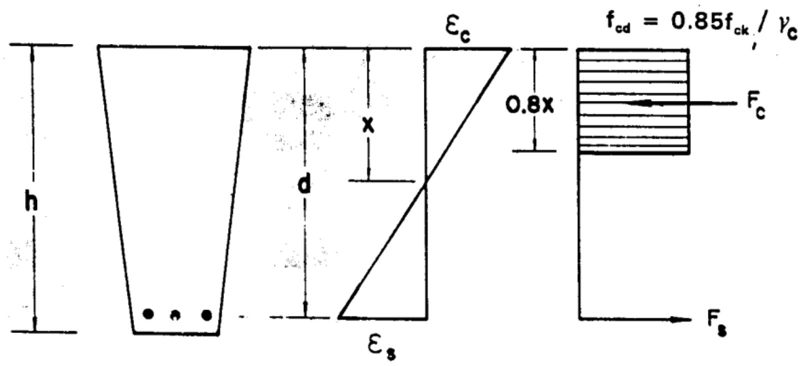


Figure 12-7 Rectangular stress block diagram

Considering the location of the neutral axis there are two genera case in reinforced concrete columns.

1. When the depth of neutral axis is within the cross section. This condition occurs when there is significant amount of moment.

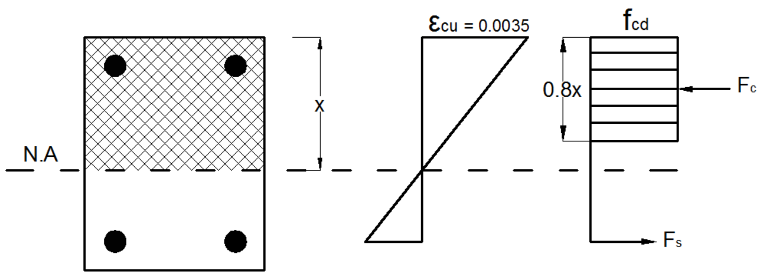


Figure 12-8 Case when the neutral axis is within the cross section

2. When the neutral axis is out the cross section. This usually occurs for very small eccentricity.

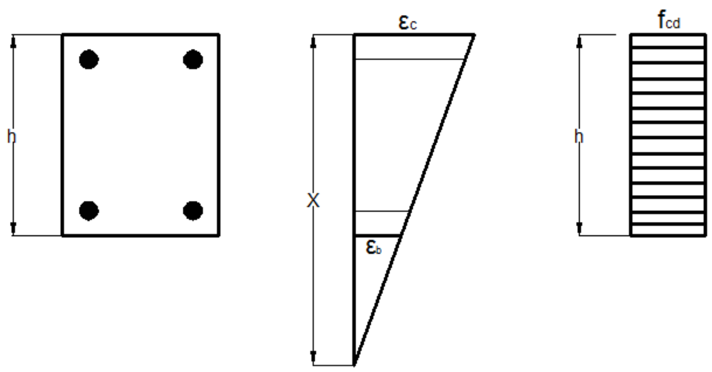


Figure 12-9 Case when the neutral axis is out of the cross section

From Figure 12-10 we categorize such cases in zone-5 where the section is subjected to predominantly compressive load. The strain in the top most compression fiber from strain for pure axial action ε_c= 0.002 up to maximum compressive strain of concrete εcu = 0.0035, keeping the distance of point C from top compression fiber at $\frac{3}{7}h$.

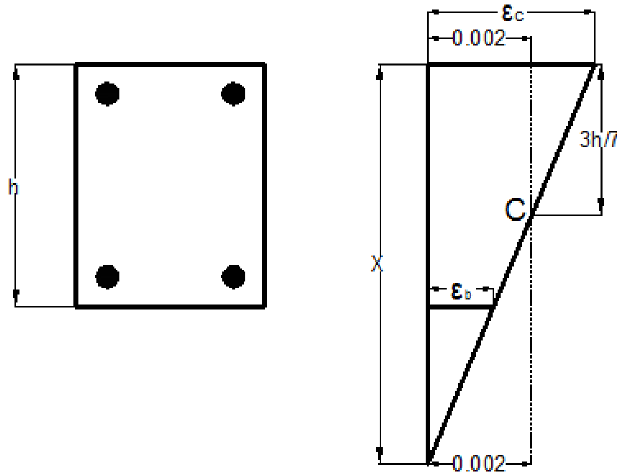


Figure 12-11 Strain in top compression fiber when the neutral axis is out the cross section

From similarity of trials we have

$$\frac{\varepsilon_c - 0.002}{\frac{3}{7}h} = \frac{0.002}{x - \frac{3}{7}h}$$

Solving for ε_c we get

$$\varepsilon_c = \frac{0.002x}{x - \frac{3}{7}h}$$

To give genera description about design of reinforced columns, consider the following symmetrically reinforced column section of with combined action of flexure and bending.

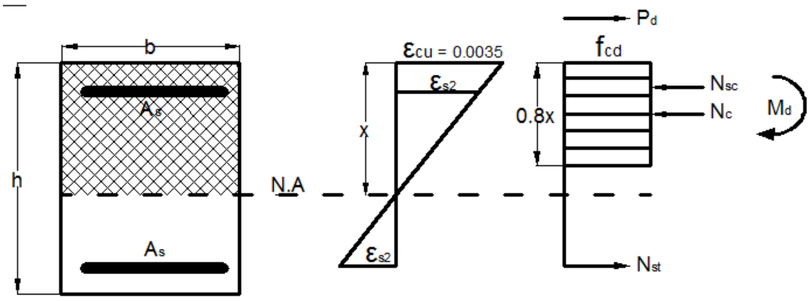


Figure 12-12 General stress strain diagram of uniaxial section with detailing type Type-1

$$P - N_c - N_{sc} + N_{st} = 0$$

$$P = N_c + N_{sc} - N_{st} \qquad (4\text{-}1)$$

Taking the summation of moments about the centroid we have.

$$\sum M = 0$$

$M = N_c\left(\frac{h}{2} - \frac{a}{2}\right) + N_{sc}\left(\frac{h}{2} - h'\right) + N_{st}\left(\frac{h}{2} - h'\right)$ (4-2) where,

N_c: Compressive force carried by the concrete.

N_{st}: Tensile force carried by the steel.

N_{sc}: Compressive force carried by the steel.

$N_c = f_{cd}(ab - A_s)$

$N_{st} = f_{st}A_s$

$$N_{sc} = f_{sc}A_s$$

The design procedure involves two basic unknowns, depth of neutral axis and area of steel which will make $P \approx P_d$ *and* $M \approx M_d$ within a certain degree of precision. For other detailing types the relationship between the unknown parameters will be complicated. To solve this problem we have two approaches.

1. **Solving the Equation Explicitly**: Solving for x from Eq. (4-1) and substituting it in the second moment equation Eq. (4-2). But, this method has a major drawback because it is difficult to know the yield state of both compression and tension steel before calculating depth of neutral axis. If we tried to express the stress in steels as a function of depth of neutral axis, we will not have one explicit relation of neutral axis depth and stress.

2. **Solving Iteratively**: Using this method firs we assume initial area of steel. Then, we take arbitrary depth of neutral axis. Using the assumed area of steel and arbitrary depth of neutral axis we calculate the magnitude of P and M and compare the result with the applied P_d andM_d. If the calculated values are different from the applied design loads we change area of steel and depth of neutral axis according utile satisfactory result is gained.

As a basis for our software development we use the second approach (Iterative Approach) with some numerical methods for rapid convergence. Detail about Iterative approach will be discussed under design procedure of uniaxial and biaxial column.

12.2. EBCS Requirements

In this topic we will list requirements of EBSC, which are directly related to reinforced concrete columns. Most of them are detailing requirements, while some others are general.

G. Minimum Size of Column

According to Art 7.2.4.1(1) the minimum lateral dimension of the column is limited to 150mm.

H. Longitudinal Bars

✓ Area of longitudinal bars should satisfy the following requirements. Art. 7.2.4.2(2).

$$A_{s,max} = 0.08A_s$$

$$A_{s,min} = 0.008A_s$$

- ✓ Minimum number of longitudinal reinforcement shall be of 4 bars in rectangular arrangement and 6 bars in circular arrangement Art. 7.2.4.2(2).
- ✓ Diameter of longitudinal bar shall not be less than 12mm.

Requirements related with lateral reinforcement are not dealt because our software only designs the cross section for flexure action only.

12.3. Method of Implementation

This topic will introduce and explain approaches that we have developed for simplification of the design procedure. The internal working principle of the software is based on point reinforcements arranged in defined coordinate system. This will make the whole procedure structured and reduce confusions.

12.3.1. Discretized Reinforcement Arrangement

Discretized reinforcement arrangement means point reinforcements (bars[1]) are placed at specific point in the coordinate system of the section. And any uniformly distributed reinforcement is changed to series of point reinforcement distributed evenly in the region. Point reinforcements should satisfy the following requirements.

1. Each unit of reinforcement is assumed to have constant level of strain in its region.
2. Each unit of reinforcement has defined area, even if they are considered to be a point.
3. Each unit of reinforcement should have defined location in the coordinate system of the section.

When coming to uniformly distributed reinforcement arrangement like in the figure_____. We approximated it by finite number of point reinforcement distributed evenly in the segment. Generally, uniformly distributed reinforcement is approximated by 32 point reinforcements distributed uniformly. In EBCS-2 Part-2, to prepare design charts the same number bars are used for uniformly distributed reinforcement. According to ACI 16 bars in rectangular arrangement are considered as uniformly distributed reinforcement.

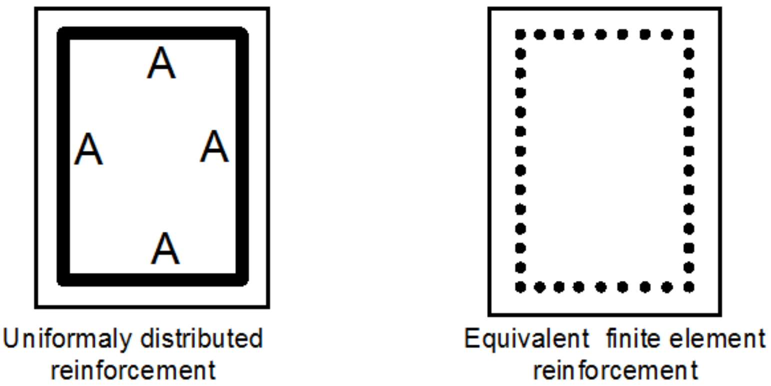


Figure 12-13 Finite element representation of uniformly distributed reinforcement

[1] Point reinforcement sometimes does not mean bar it can also be a bundle of bars.

If the neutral axis for the cross section is parallel with uniformly distributed reinforcement the whole segment can be approximate by single point reinforcement located at its centroid. For example in uniaxial column with detailing type shown in bellow the reinforcement is represented by point reinforcement located at its middle.

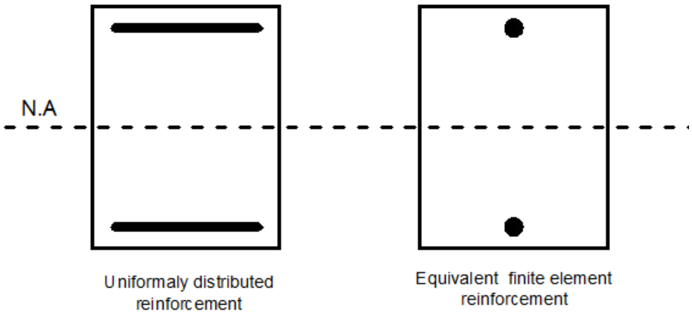


Figure 12-14 Uniformly distributed reinforcement representation when the neutral axis is parallel with the reinforcement segment considered.

Note that, discretization does not mean actual detailing or placing of bars. It only represents design simplification for modeling the section in order to calculate area of reinforcement required. It is only considered for implementing the design procedure in computer program.

12.3.2. Coordinate System

The coordinate of each reinforcement unit is measured from the centroid of the column section. In addition the moments in x and y direction are calculated about this origin. This assumption is taken to simplify the moment equation making, so that the applied axial load will not make force from the centroid of the cross section. The same coordinate system also applicable for moment carried by the concrete.

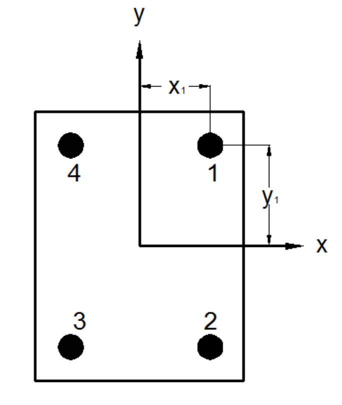


Figure 12-15 Coordinate system of reinforcement units

12.3.3. Filling Coordinates of Reinforcement Units

If detailing is not done, the user is expected to provide cover to reinforcements manually. This is done by specifying b' and h'. But if detailing is required to be don, first we assume initial bar diameter $\emptyset = \emptyset_{min}$ from the bar preference of the user. Then, by adding the concrete cover, stirrup diameter and half of the $\emptyset_{min}$ we calculated h' and b' for both uniaxial and biaxial columns.

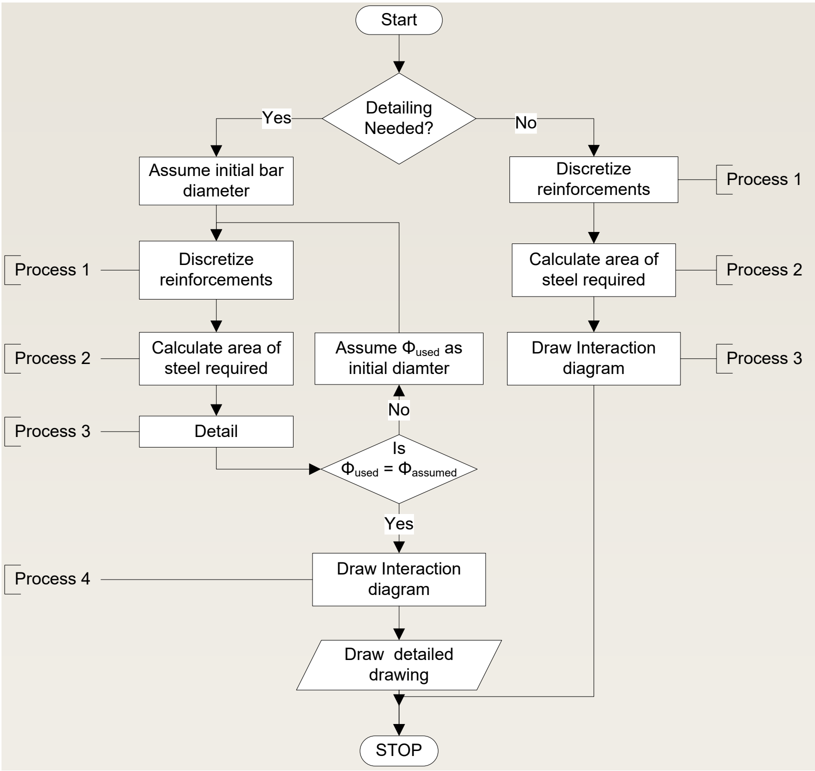
$$h' = b' = cover + \frac{\emptyset_{\text{stirrup}}}{2} + \frac{\emptyset_{min}}{2}$$

Now once $h'\ and\ b'$ are calculated the next step is to fill the coordinates of all the reinforcement units from the origin of the coordinate system.

12.4. Design of Uniaxial Column

Design of uniaxial column starts by placing of reinforcements with initially assumed area of steel. Then, the section capacity is calculated and checked with the applied design load. If the provided area of steel is insufficient to support the loads, reinforcement area is changed and the design capacity is checked with the applied loads. This process is repeated until the area of reinforcement is sufficient enough to support the applied loads. Finally if the user needed for detailing to be done the calculated area of reinforcement furnished with bars.

In the following subsequent topics each steeps in the design procedure are discussed in paragraphs and finally represented by stepwise algorithm and flow charts. Generally, the design uniaxial column for this software component goes according to the following flow chart.



Flow Chart 12-1

Process labeled as 1,2,3 and 4 will be shown in detail in the following section.

12.4.1. Area of Reinforcement for Pure Axial Action

For columns supporting only axial load the minimum are of reinforcement required to support the applied load is calculated solving for area of steel from the force equilibrium equation of the section. The maximum strain in concrete under pure axial load is given to be 0.002. From the force equilibrium we have

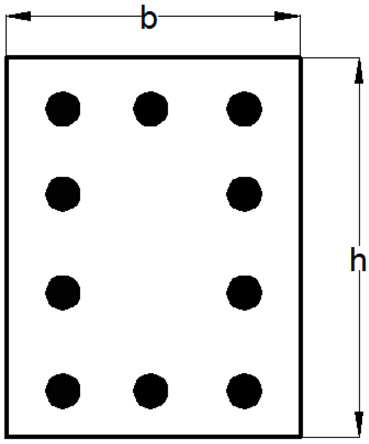


Figure 12-16

$$P_d = f_{cd}\left(bh - A_{s,total}\right) + f_s A_{s,total}$$

Before calculating area of steel we need to check the yield state of the steel. This is required because the strain in concrete is limited to 0.002 and the steel also should have the same strain with the surrounding concrete. If the design yield strain of the steel is below 0.002 we take f_s equal with f_{yd}. Otherwise, we take $f_s = 0.002E_s$.

$$f_s = \begin{cases} f_{yd}, & \varepsilon_{yd} \leq 0.002 \\ x, & \varepsilon_{yd} = 0.002 \end{cases}$$

$$A_{s,total} = \frac{(P_d - f_{cd}bh)}{f_{yd} - f_{cd}}$$

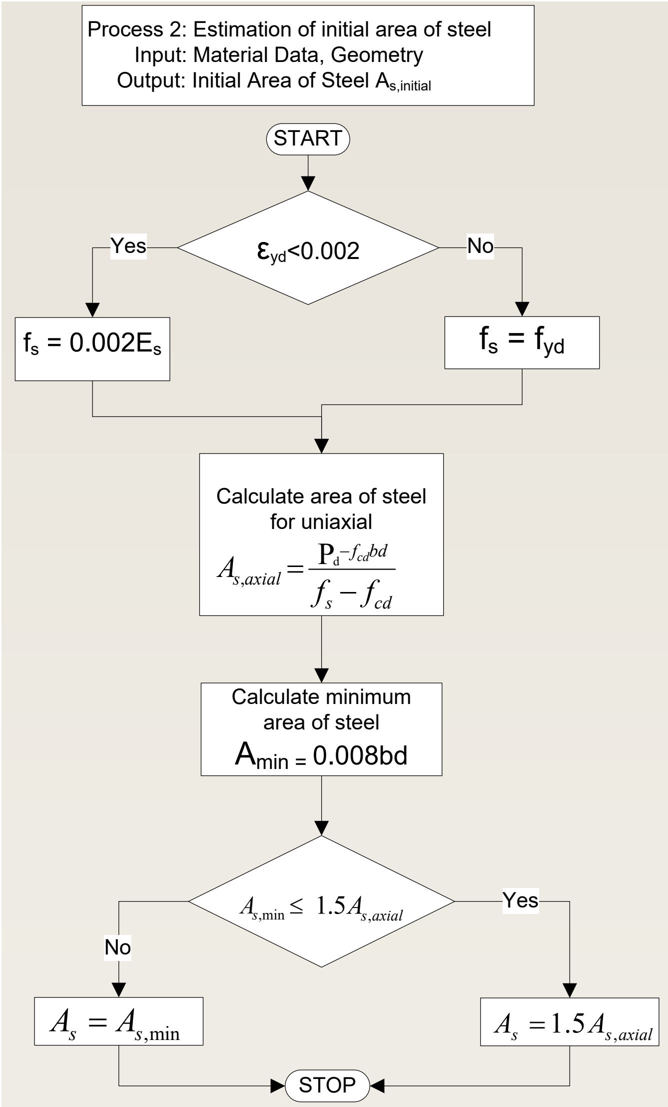
12.4.2. Estimation of Initial Area of Reinforcement

To begin the design we need to assume initial area of steel. This initial area of steel can be taken as maximum of 1.5 time area of steel required for pure axial action or minimum reinforcement requirement. The area of steel required for pure axial action is calculated as per Section 4.5.1.

$$A_{s,initial} = max\{\, A_{s,min}\,, 1.5A_{s,axial}\}$$

$A_{s,axial}$: Area of steel required for pure axial load P_d.

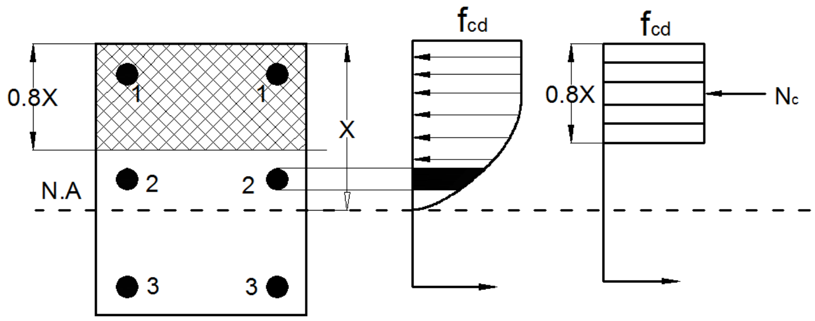
Factor 1.5 is considered arbitrarily to roughly account the effect of bending. See the following flow char.



Flow Chart 12-2 Estimation of initial area required for the section

12.4.3. Force and Moment Carried by the Concrete

For a given depth or neutral axial and area of reinforcement force carried by the concrete is calculated using rectangular stress block diagram described in Section 4.2. First, we calculate area of concrete above depth of equivalent rectangular stress block a. When calculating this we should subtract the area of reinforcement in the compression zone. Usually, when using rectangular stress block reinforcement above 0.8x is subtracted. But, in ESADS we subtracted the area of all reinforcements located above the neutral axis. The reason for this is the effect of the reinforcement area when considering the parabolic stress distribution. In other word, reinforcement units located bellow 0.8x and above x have effect on the parabolic stress distribution, but; they have no effect on the rectangular stress block. Therefore, to be safe we subtracted the area of all reinforcements above the neutral axial. See the following figure for more detail.



In the above figure if we consider rectangular stress block only bar 1 will be subtracted, but if you consider the parabolic stress diagram both of bar 1 and 2 are subtracted from the area of concrete in the compression zone.

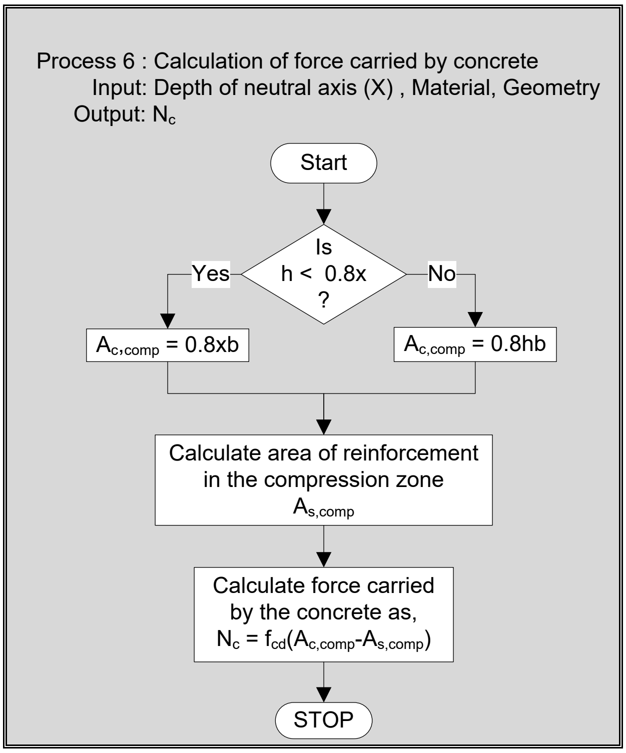
Considering point reinforcement arrangement, the force and moment carried by the concrete is computed as,

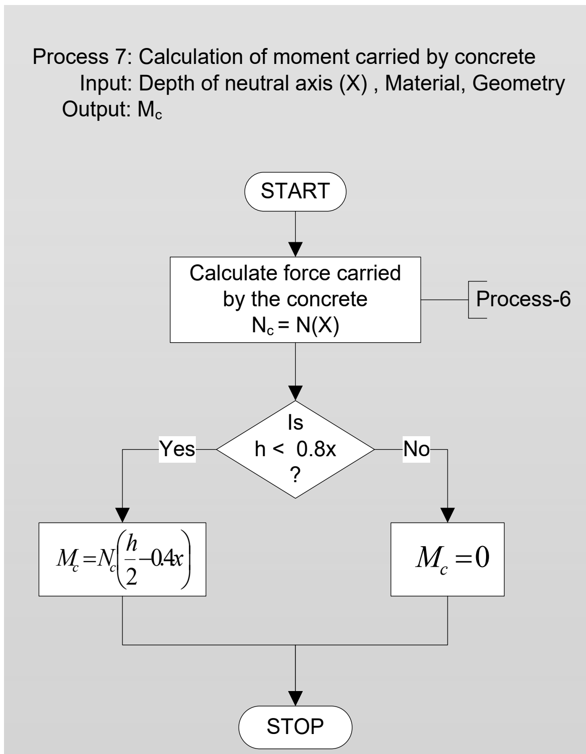
$$N_c = f_{cd}(A_{c,comp})$$

$$M_c = f_{cd}\left(A_{c,comp}\right)\frac{a}{2}$$

$$A_{c,comp} = 0.8xb - \sum_{i=1}^{n} A_{si,comp}$$

$\sum_{i=1}^{n} A_{si,comp}$: Represents the sum of area of bars located in the compression zone.





Flow Chart 12-3 Force and moment carried by the concrete

12.4.4. Force and Moment Carried by Reinforcements

Force carried by the steel is the sum of force carried by each reinforcement unit. As we have described in Section 4.4.1, all reinforcements are converted to point reinforcements having defined strain, stress and area. This greatly simplifies and generalizes the algorithm.

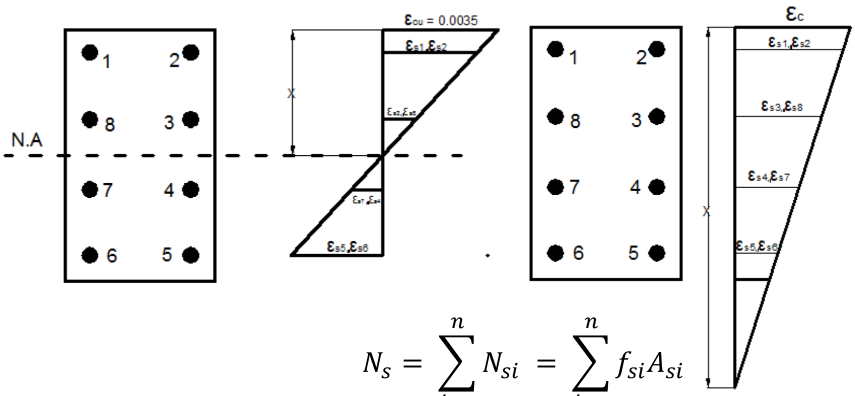
The first step in calculating the force carried by reinforcements is to calculate the strain level of the top compression fiber. While doing so, we have two general cases described in Section 4.2.

When the neutral axis is within the cross section we take ε_{cu} = 0.0035 and when it is outside the section we take $\varepsilon_c = \frac{0.002x}{x-\frac{3}{7}h}$.

Once the strain in the top compression fiber is determined, the next step is to calculate the strain in each reinforcement unit and their corresponding stress. Then, the force in each reinforcement is calculated by multiplying the stress by their corresponding area.

Using the calculated force in the reinforcements we calculate the moment carried by each area of steel by multiplying the force by the moment arm taking the centroid of the section as our origin.

Finally, superposing force carried by each reinforcement unit will give the total force carried by the reinforcements. In a similar manner, superposing the moment carried by each reinforcement unit gives the total moment carried by reinforcements



$$N_s = \sum_{i=1}^{n} N_{si} = \sum_{i=1}^{n} f_{si}A_{si}$$

$$M_s = \sum_{i=1}^{n} M_{si} = \sum_{i=1}^{n} f_{si}A_{si}y_{si}$$

$$f_{si} = \begin{cases} f_{si}, & |\varepsilon_{si}| \geq \varepsilon_{yd} \\ E_s\varepsilon_{si}, & |\varepsilon_{si}| < \varepsilon_{yd} \end{cases}$$

$$\varepsilon_{si} = \varepsilon_{cu}\frac{y_i + x - \frac{h}{2}}{x}$$

N_s: Total Force carried by reinforcements.

N_{si}: Force carried by reinforcement unit i.

M_s: Total moment carried by reinforcements.

M_{si}: Moment carried by reinforcement unit i.

n: Total number of reinforcements.

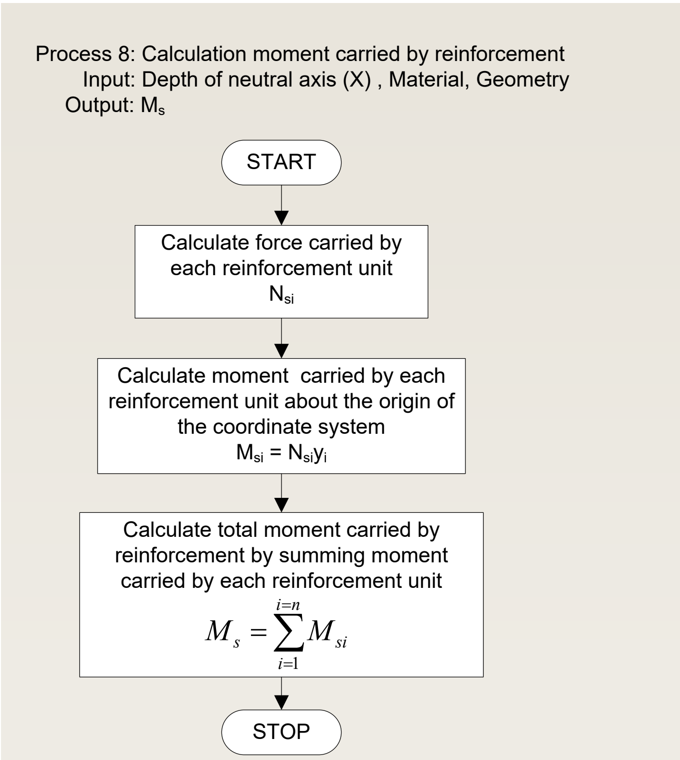
f_{si}: stress in reinforcement unit i.

ε_{si}: strain in reinforcement unit i.

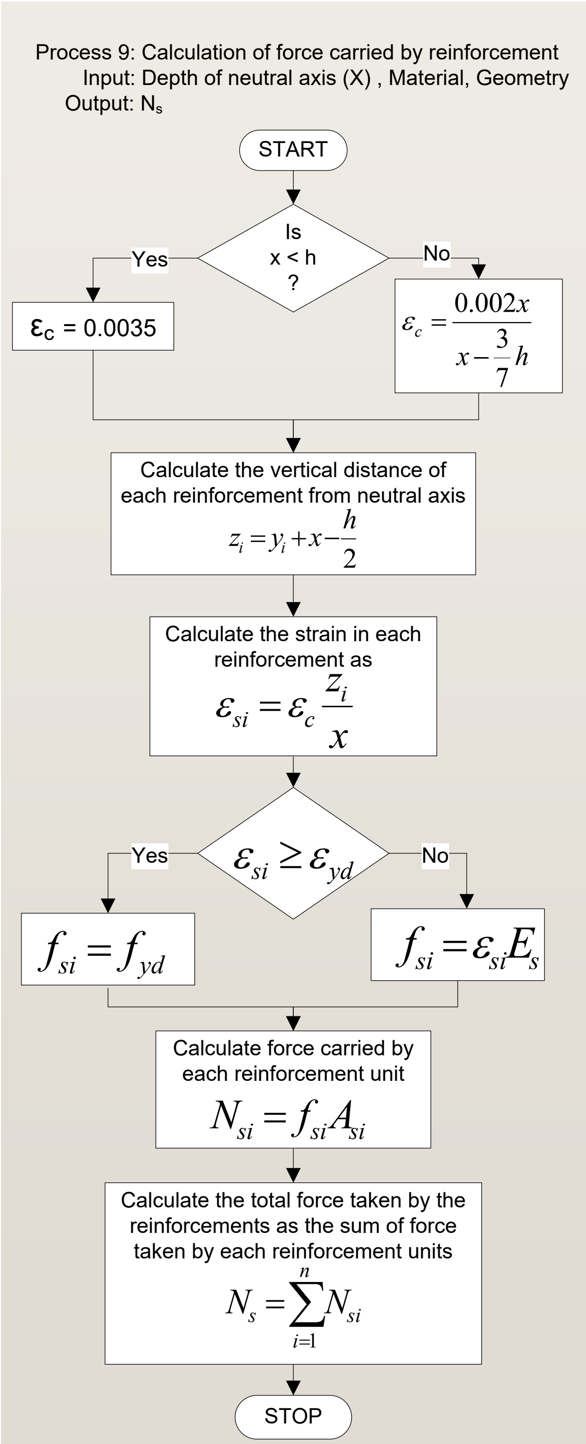
A_{si}: Area of reinforcement unit i, in our case constant for all units.

y_i: The y-coordinate of the reinforcement measured from the geometric centroid of the section.

Note that, in the formula we are not required to subtract the force carried by compression steel separately, the strain account it being negative in the compression zone. See the following two flow charts for force and moment carried by the reinforcement.



Flow Chart 12-4 Moment carried by the steel.

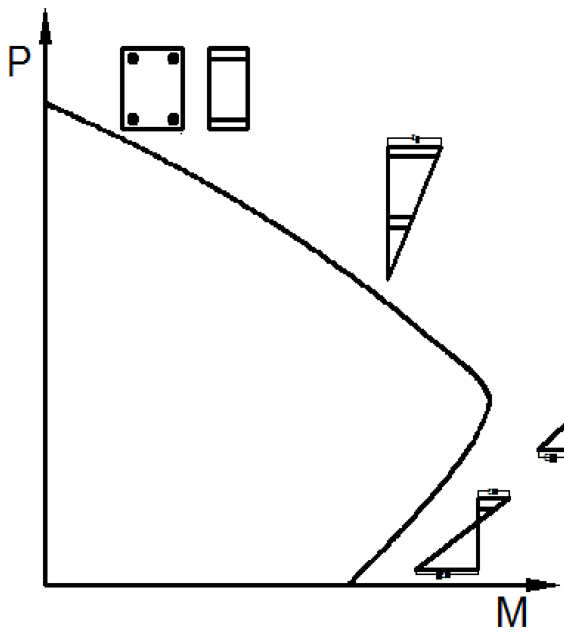


Flow Chart 12-4 Force carried by the steel

12.4.5. Depth of Neutral Axis for a Given Axial Load

Usually, we need depth of neutral axis, which will give the required axial load resistance. To get the exact value of x the given axial load P should be less than the pure axial capacity. Otherwise, x will not have solution.

The following interaction diagram shows the variation of depth of neutral axis, axial load and moment.



The axial load (P) varies nonlinearly with respect to x, depending on yielding and unyielding of reinforcing units and location of neutral axis i.e. inside the cross section or outside the cross section. In Section 4.2, we noted that the use of numerical or iterative approach is better to implement the procedure in computer. There are five numerical methods, which are used to solve nonlinear equations. These are,

1. Bisection Method
2. Regula Falasi Method
3. Secant Method
4. Fixed Point Iteration Method
5. Newton Raphson Method.

The last two methods need explicitly expression of the function under consideration. But, in our case there is no one explicit mathematical function, which represents the variation of P and x. Therefore, we are forced to focus on the reaming three methods.

Bisection method always converges but it needs many number of iteration to get the exact solution. Secant method has higher degree of convergence but, it diverges in some special cases. The iteration will diverge when the approximating secant line comes horizontal. But, if we consider Regula Falasi Method it have higher rate convergence relative to bisection method and converges always because it is developed on the basis of intermediate value theorem. Concerning the detail of each numerical method the reader should refer to specialist literatures on numerical methods. Some references are list at the end of this chapter.

Thus, we choose Regula Falasi method to be the best to fit our scenario. Mathematically we can express the relationship between P and x as

$$P = N_c(x) + N_s(x)$$

$$N_c(x) + N_s(x) - P = 0$$

$N_c(x)$, force carried by the concrete is calculated based on Section 4.5.3 using x as a depth of neutral axis. And $N_s(x)$, force carried by the steel is calculated based on Section 4.5.4

First, assume two initial values of x say x_1 and x_2, which satisfies the following requirement of Regula Falasi Method.

$$P_1 = N_c(x_1) + N_s(x_1) < P$$

$$P_2 = N_c(x_2) + N_s(x_2) > P$$

This requirement is needed because Regula Falasi method is based on Intermediate Value Theorem[2] for the purpose of unconditional convergence. From the above two equation we can conclude that there exists depth of neutral axis x between x_1 and x_2, which gives

$$P = N_c(x) + N_s(x)$$

For the first iteration, the value of x is estimated as

$$x_{new} = \frac{P-P_1}{P_2-P_1}(x_2 - x_1) + x_1$$

And the corresponding value axial load resistance at $x = x_{new}$, P_{new} is determined as

$$P_{new} = N_c(x_{new}) + N_s(x_{new})$$

Now we have to change the value of x_1 and x_2 for the next iteration depending on the newly computed value of x_{new} and P_{new}.

If $P_{new} < P$

$x_1 = x_{new}$

$P_1 = P_{new}$, while P_2 *and* x_2 remain unchanged.

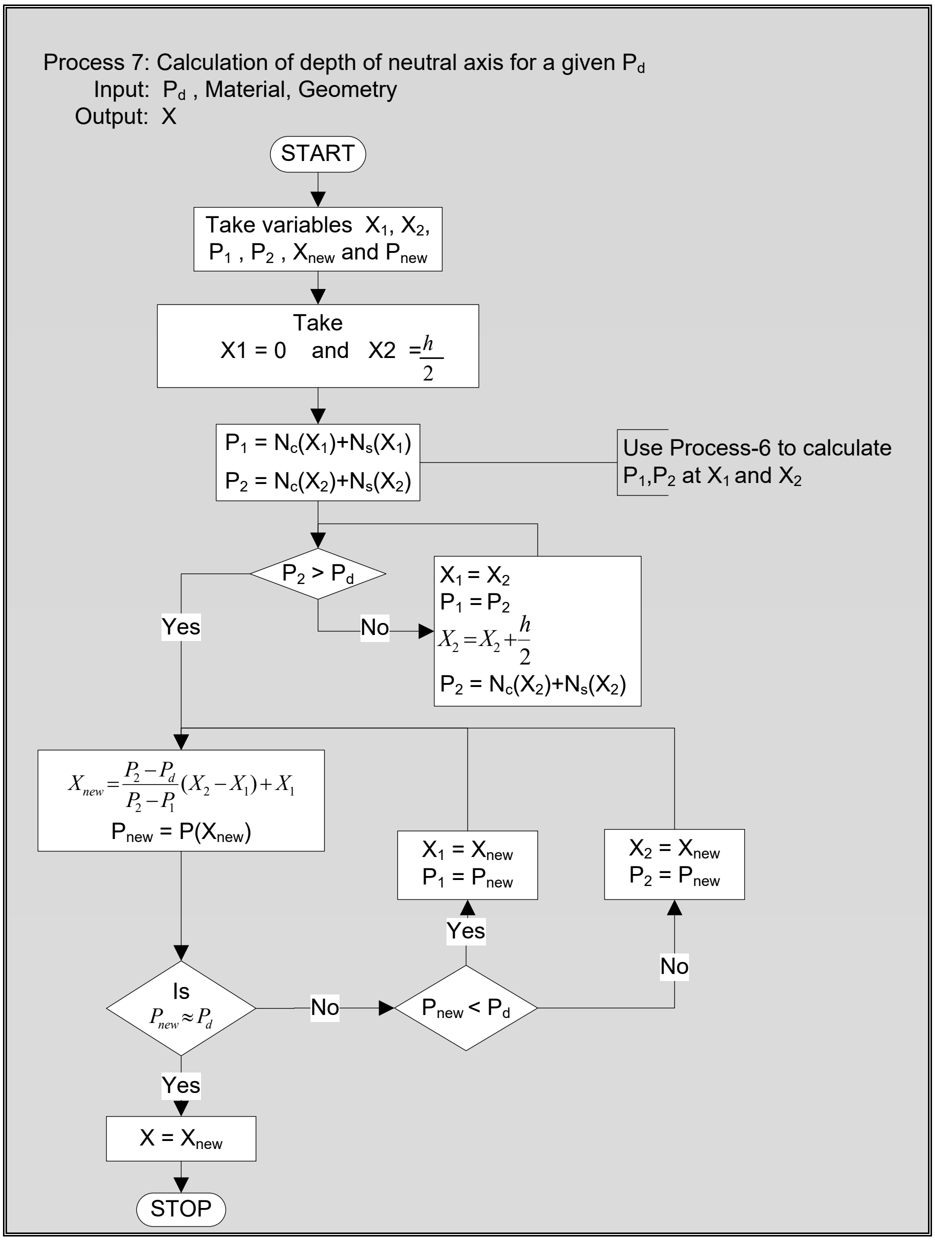
Else if $P_{new} \geq P$

$x_2 = x_{new}$

$P_2 = P_{new}$, while P_1 *and* x_1 remain unchanged.

Finally, using the newly computed value of x_1, x_2, P_1 *and* P_2 the iteration is proceeded until P_{new} is approximately equal with P within a certain degree of precision. In this software to end the iteration 3 decimal places accuracy is needed for P_{new} in KN unit. The following flow char summarize all the steps involved.

[2] If two point in a curve are located below and above x axis, then we can say that the curve crosses the x axis at least once between these two points.



Flow Chart 12-5 Calculation of depth of neutral axis

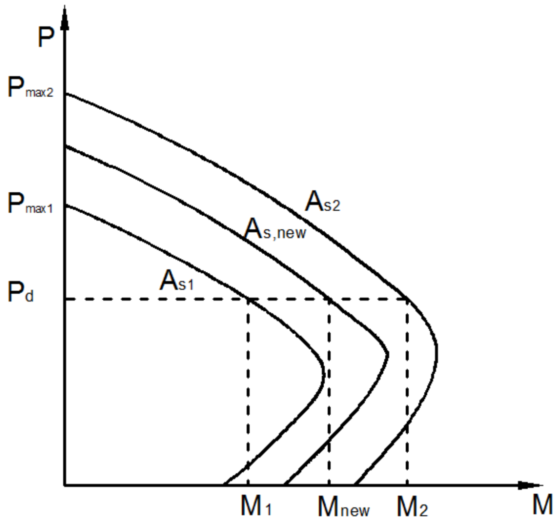
12.4.6. Calculating Area of Reinforcement Required

The main procedure the whole design process is calculating the minimum reinforcement area required to carry the applied axial load P_d and moment M_d. To compute this required area of reinforcement first we need to assume two initial area of reinforcement for the cross section. The first initial area of reinforcement A_{s1} is calculated using the principle given under Section 4.5.2 and the second area of steel A_{s2} is taken as the A_{max}. Then, using the initially assumed area of reinforcements, we calculate the value of neutral axis depths x for both reinforcements at axial load of P_d as descried in Section 4.5.5. Taking the calculated values of x we compute the moment capacity of the cross section for both reinforcements. This can be done by adding moment carried by concrete described in Section 4.5.3 and moment carried by the steel described in Section 4.5.4.

$$A_{s1} = max\,\{A_{s,min}\,, A_{s,axial}\}$$

$$A_{s2} = A_{s,max}$$

Let's denote moment resisted by A_{s1} as M_1 and by A_{s2} as M_2 drawing interaction diagram for both areas of reinforcements we get the following diagram.



Now if $M_2 < M_d$ the section is considered as over reinforced and when $M_1 > M_d$,while $A_{s1} = A_{min}$ the section is considered as under reinforced and the design is terminated.

Using A_{s1}, A_{s2} ,M_1 and M_2 the new better approximate of area of steel is made by linear interpolation of the previous two values.

$$A_{s,new} = \frac{M_2 - M_d}{M_2 - M_1}(A_{s2} - A_{s1}) + M_1$$

For the next iteration, using the calculated new area of reinforcement $A_{s,new}$ we calculate the new moment capacity at the axial load of P_d. As usual first calculate x using $A_{s,new}$ at axial load of P_d as described in Section 12.4.5. Then, using the computed value of x we calculate the total moment by adding moment carried by the concrete referring to Section 4.5.3 and moment carried by reinforcement referring to Section4.5.4. Denote this computed moment as M_{new}.

Finally, for the next iteration take the following values.

$$A_{s1} = A_{s2}$$

$$A_{s2} = A_{s,new}$$

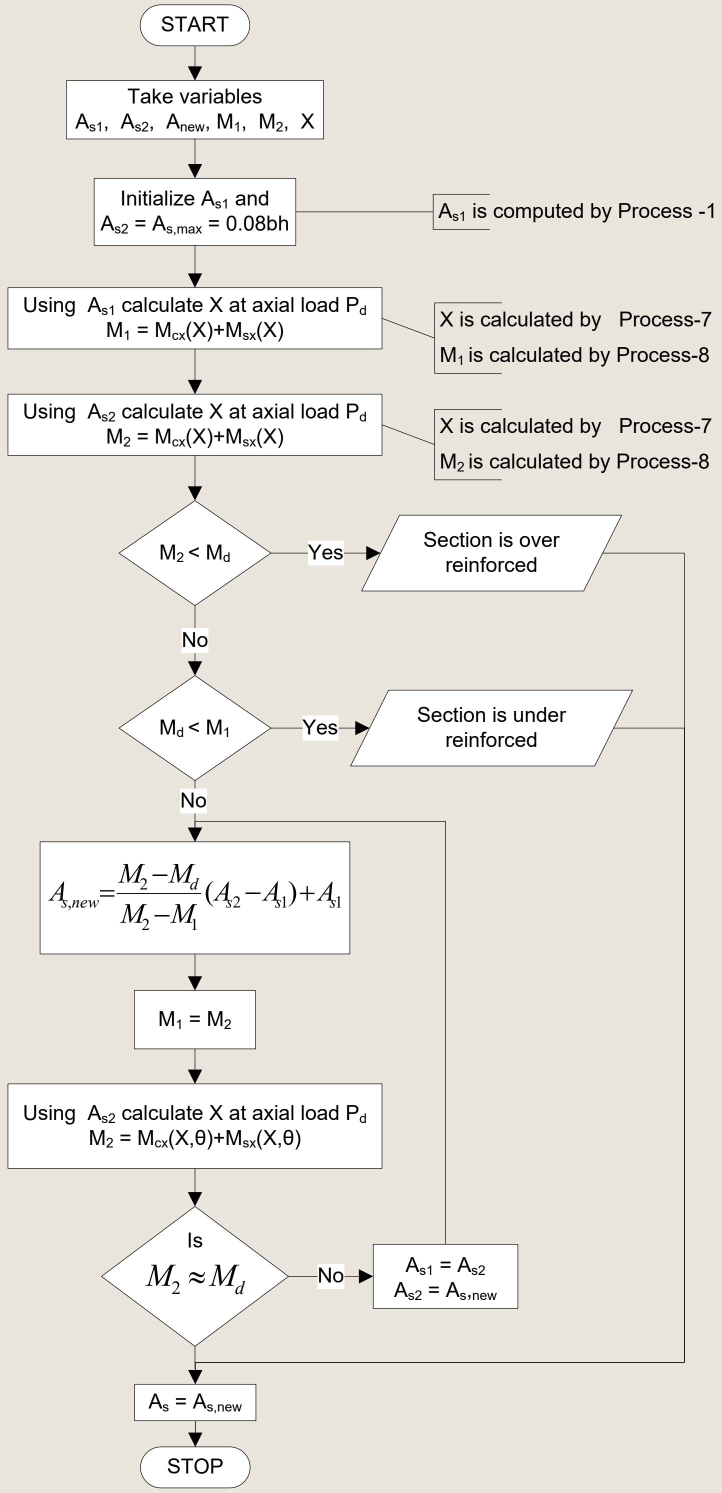
$$M_1 = M_2$$

$$M_2 = M_{new}$$

Now for better approximation of area of reinforcement the same procedure is repeated until the required degree of precision is reached. In this software the iteration is completed when M_d and M_{new} are correct to 6 decimal place in KNm unit.

Note: Implementing the algorithm in computer we have understood that the area of reinforcement and the moment capacity at constant axial load have a linear relationship. Thus, linear interpolation between successive iteration converges rapidly. Approximately, for uniaxial column, it takes only three iteration to get the design moment correct to three decimal place.

Process 2 : Calculating area of reinforcement required to carry the applied load
Input: M_d, P_d , Material, Geometry
Output: A_s



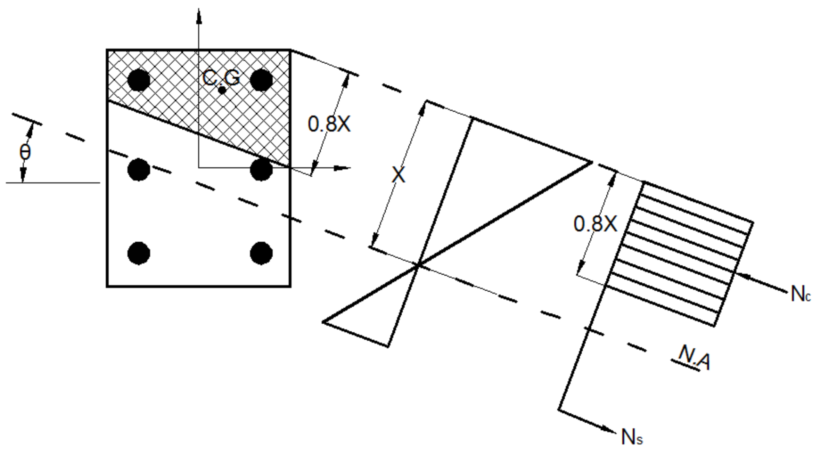
12.5. Design of Biaxial Columns

Similar to uniaxial columns, the design of biaxial column also involves initial estimate of area of steel required. Then, if there is any uniformly distributed reinforcement we discretized it as point reinforcements. Using the assumed area of reinforcement the section capacity is checked for the applied axial load P_d , M_{xd} and M_{yd}. If the assumed area of steel is sufficient to support the applied loads we end the design. Otherwise, we change the area of reinforcement until it is sufficient enough to support the applied loads i.e. P_d , M_{xd} and M_{yd}. Finally, the computed area of steel furnished as reinforcement bars in the cross section if the user needs the detailing.

Most design principles and procedures of biaxial columns are similar with uniaxial columns except they involve inclined neutral axis due to biaxial bending effect in the two principal axes. To avoid redundancy here we only focus on those procedures, which are unique to biaxial column. Therefore, the reader is expected to make a reference to uniaxial column design procedures whenever necessary.

12.5.1. Force and Moment Carried by the Concrete

This topic describe the way to calculate the force and moment carried by the concrete for biaxial column for a given depth and inclination of neutral axis. Here we need to compute area and centroid of polygonal compression zone formed as a result of inclined neutral axis. In the same way with uniaxial column the area of reinforcements above the neutral axis are subtracted from the area of compression zone. Generally, for biaxial column we have the following five types of neutral axis orientation. For all this conditions, we have derived equation which is used to calculate the area of and centroid of the shaded region.



$$N_c = f_{cd}(A_{c,comp})$$

$$M_{cx} = N_c\bar{y}$$

$$M_{cy} = N_c\bar{x}$$

N_c : Force carried by the concrete.

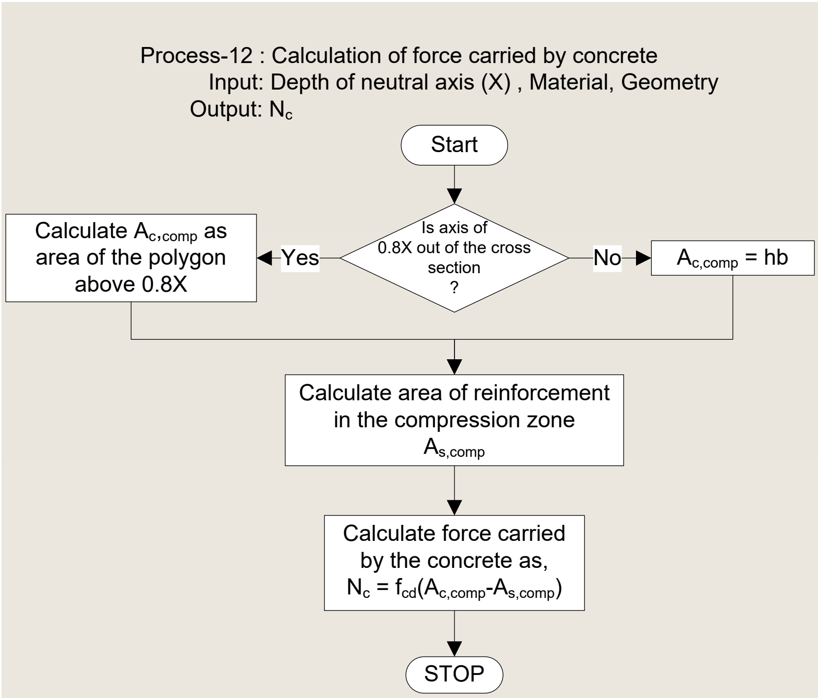
M_{cx}: Moment carried in x-direction by the concrete.

M_{cy}: Moment carried in y-direction by the concrete.

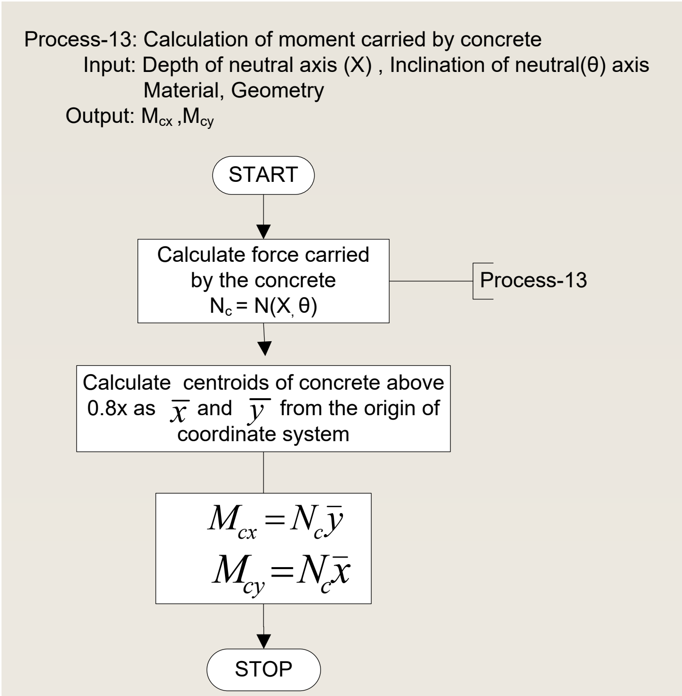
$\bar{x}$: The x-coordinate of centroid of concrete in compression zone from the centroid of the section.

$\bar{y}$: The y-coordinate of centroid of concrete in compression zone from the centroid of the section.

θ: Angle of inclination of neutral axis measured from the horizontal.



Flow Chart 12-6 Calculation of force carried by the concrete



Flow Chart 12-7 Moment carried by the concrete

12.5.2. Force and Moment carried by Reinforcements

Here the force and moment carried by the steel for a given depth and inclination of neutral axis x and θ respectively are computed in similar manner as uniaxial columns described in Section 4.5.4. The difference here is only inclined neutral axis making the computation somewhat complicated. The notes written under uniaxial column on this topic are also applicable here.

Due to the inclination of the neutral axis the strain in each reinforcement units should be computed using perpendicular distance measured from the inclined neutral axis to the centroid of the reinforcements. This perpendicular distance is denoted as z. For more detail see the previous figure. Here also there are two cases when the neutral axis is inside and outside the cross section. Once the strain in the concrete is determined we follow the same procure for both cases. Refer to the following figures for more detail.

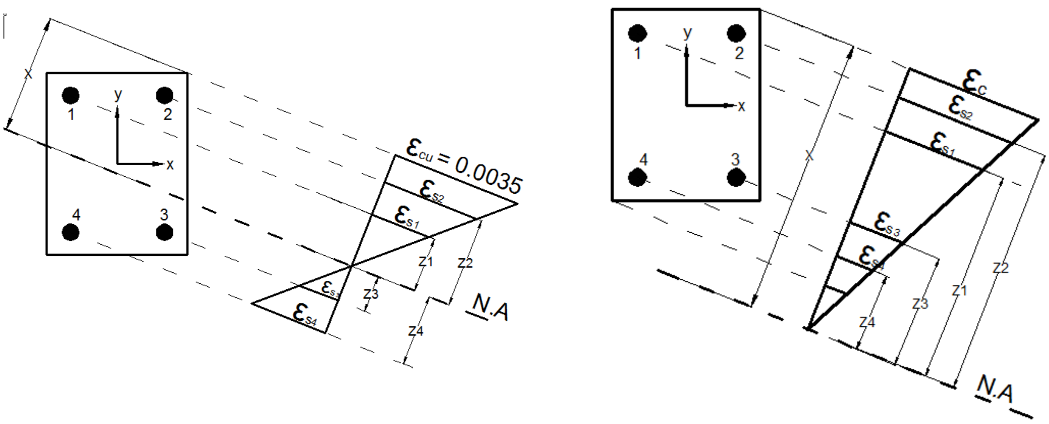


Figure 12-17 Location of neutral axis the left one: when the neutral axis is within the x-section and the right one: when the neutral axis is outside the x-section

$$N_s = \sum_{i=1}^{n} N_{si} = \sum_{i=1}^{n} f_{si} A_{si}$$

$$M_{sx} = \sum_{i=1}^{n} M_{sxi} = \sum_{i=1}^{n} N_{si} y_{si}$$

$$M_{sy} = \sum_{i=1}^{n} M_{syi} = \sum_{i=1}^{n} N_{si} x_{si}$$

$$f_{si} = \begin{cases} f_{si}, & |\varepsilon_{si}| \geq \varepsilon_{yd} \\ E_s \varepsilon_{si}, & |\varepsilon_{si}| < \varepsilon_{yd} \end{cases}$$

$$\varepsilon_{si} = \varepsilon_{cu} \frac{z_i}{x}$$

N_s: Total Force carried by reinforcements.

N_{si}: Force carried by reinforcement unit i.

M_{sx}: Total moment carried by reinforcements in x-direction.

M_{sy}: Total moment carried by reinforcements in y-direction.

M_{sxi}: Moment carried by reinforcement unit i in x-direction.

M_{syi}: Moment carried by reinforcement unit i in y-direction.

n: Total number of reinforcements.

f_{si}: Stress in reinforcement unit i.

ε_{si}: Strain in reinforcement unit i.

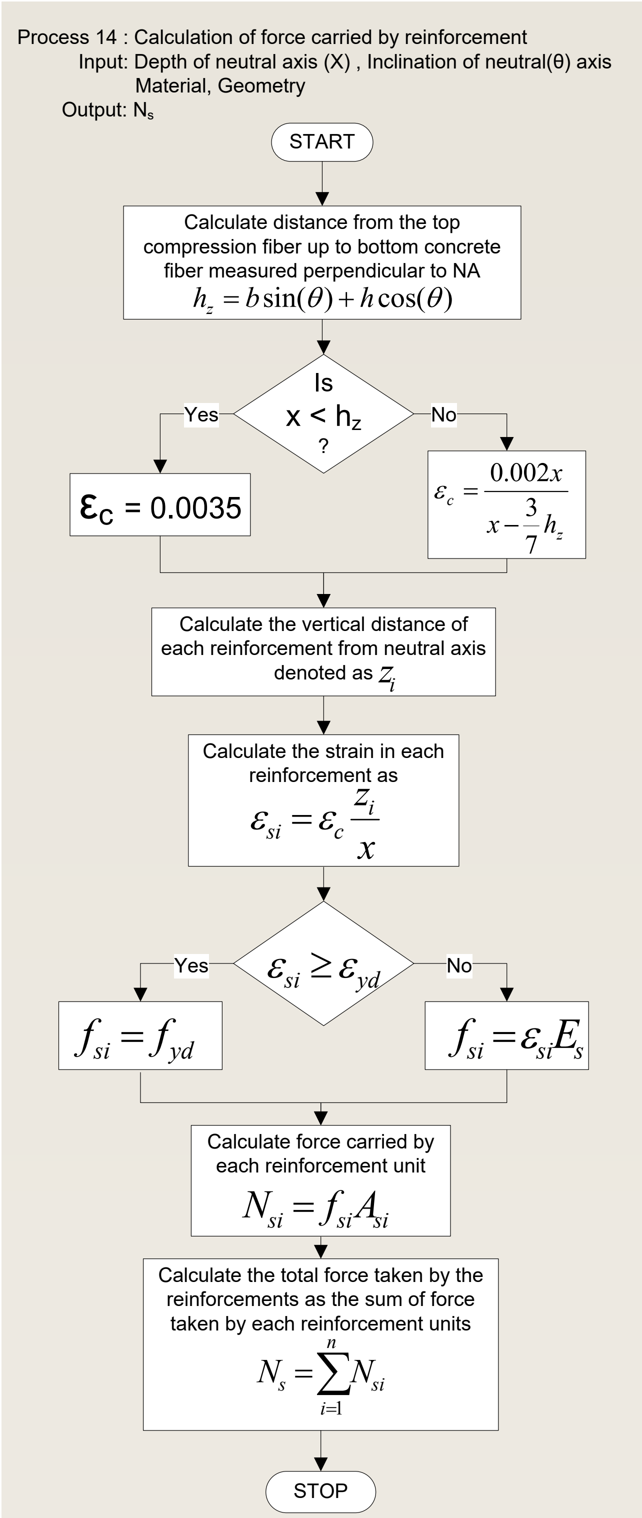
A_{si}: Area of reinforcement unit i, in our case constant for all units.

y_i: The y-coordinate of the reinforcement measured from the geometric centroid of the section.

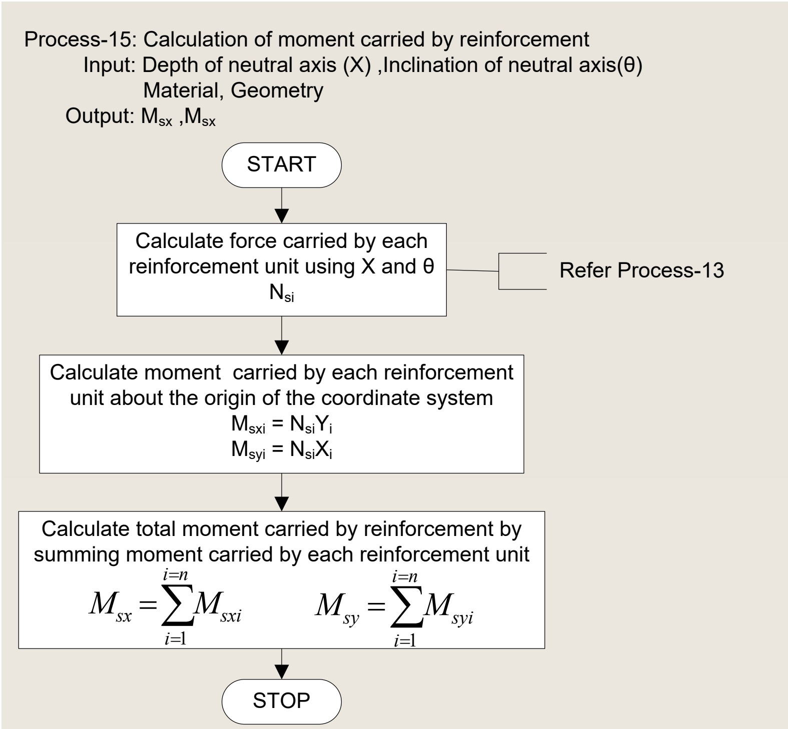
x_i: The x-coordinate of the reinforcement measured from the geometric centroid of the section.

z_i: The perpendicular distance from the inclined neutral axis to centroid of reinforcement i.

θ: Angle of inclination of neutral axis measured from the horizontal.



Flow Chart 12-8 Calculation of force carried by the steel



Flow Chart 12-9 Moment carried by the reinforcements

12.5.3. Calculating Area of Reinforcement Required

Calculation of reinforcement required for biaxial column section involves three unknowns i.e. depth and inclination of neutral axis (x and θ) and area of reinforcement itself. As uniaxial column we also approach this problem iteratively. We assume two initial area of reinforcement. The first area of reinforcement A_{s1} is calculated using the principle given under Section 4.5.2 and the second area of steel A_{s2} is taken as the A_{max}. Then, using these assumed areas of reinforcements, we calculate the value of neutral axis depths x and θ which will give an axial resistance of P_d ,while M_x/M_y ratio being M_{xd}/M_{yd}. Taking the calculated values of x and θ, we compute M_x and M_y resistance of section for both reinforcements. This can be done by adding moment carried by concrete described in Section 4.6.1 and moment carried by the steel described in Section 4.6.2. If the calculated M_x and M_y are approximately equal with the design values M_{xd} and M_{yd} we end the design. Otherwise, by changing the area of reinforcement we repeat the same procedure until the calculated M_x and M_y are equal with the applied loads to a certain degree of precision.

The following interaction surface shows the procedure of calculating reinforcement pictorially.

Now if $M_{x2} < M_{xd}$ at $\frac{M_{x2}}{M_{y2}} = \frac{M_{xd}}{M_{yd}}$ the section is considered as over reinforced and when $M_{x1} > M_{xd}$ at $\frac{M_{x1}}{M_{y1}} = \frac{M_{xd}}{M_{yd}}$, while $A_{s1} = A_{min}$ the section is considered as under reinforced and the design is terminated.

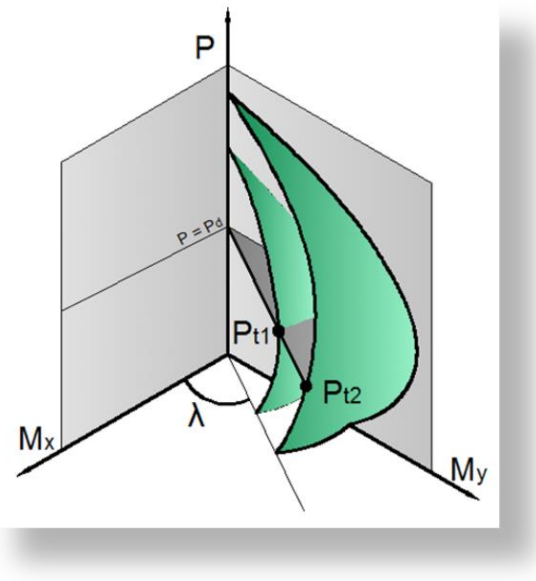


Figure 12-18 Brocken-out section showing interaction surface drawn for two different area of reinforcements.

Similar to uniaxial columns, here also the area of steel and moment in one of the axes at constant axial load and $\frac{M_x}{M_y}$ ratio approximately has linear relationship. Therefore, successive values of area of reinforcement are calculated as a linear interpolation of the previous two values of moment and area of steel in the previous iteration.

From the interaction surfaces drawn in figure____ we have two point P_{t1} and P_{t2}, which are on the interaction surfaces drawn based on A_{s1} and A_{s2} respectively.

$P_{t1} = (P_d, M_{x1}, M_{y1})$ Drawn for A_{s1}

$P_{t2} = (P_d, M_{x2}, M_{y2})$ Drawn for A_{s2}

Using linear interpolation the new better approximate of area of reinforcement is computed as,

$$A_{s,new} = \frac{Mx_2 - M_{xd}}{M_{x2} - Mx_1}(A_{s2} - A_{s1}) + M_{x1}$$

Alternatively, the interpolation also can be done using M_{y1} *and* M_{y2}, the result will be the same since M_{y1} *and* M_{y2} are constant multiple of M_{x1} *and* M_{x2}. Finally, for the next iteration we take,

$A_{s1} = A_{s2}$

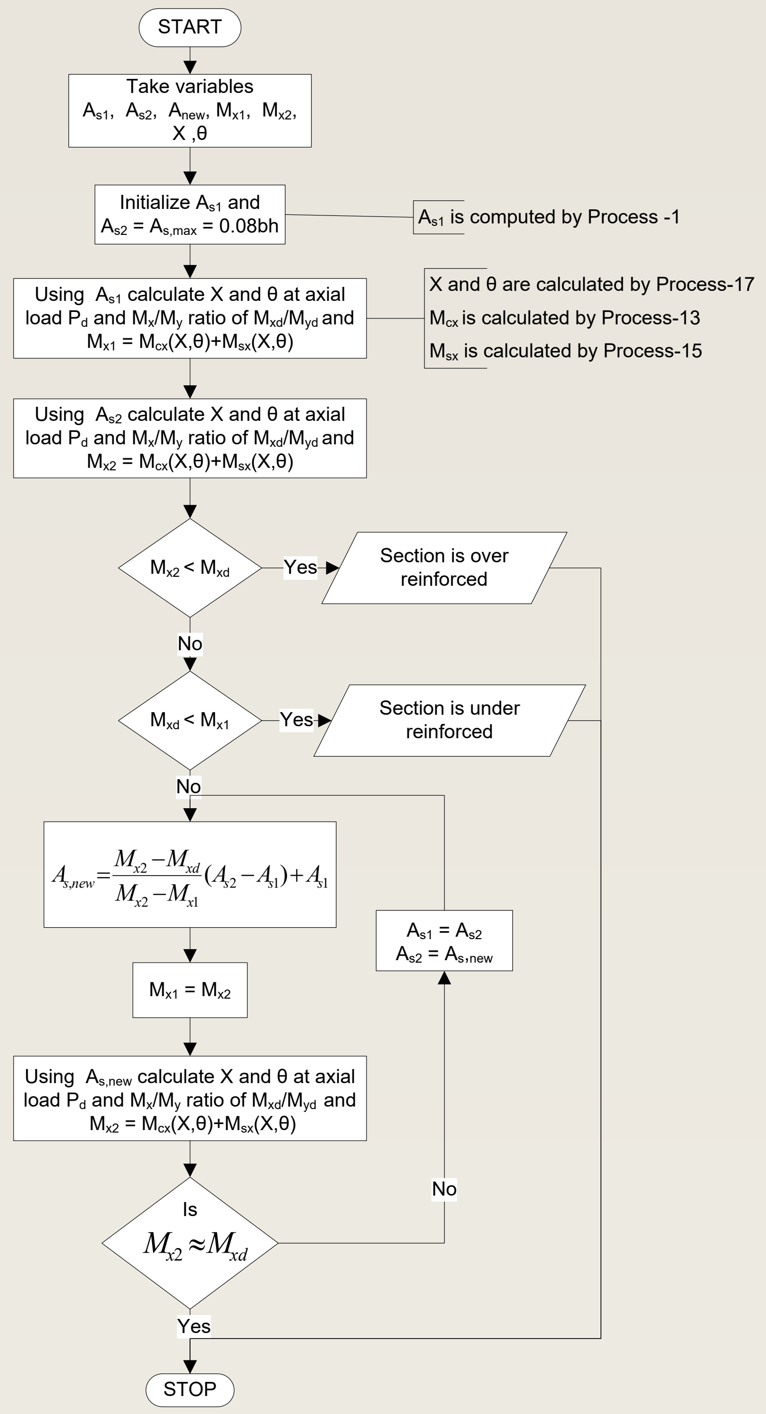
$A_{s2} = A_{s,new}$

$M_{x1} = M_{x2}$

$M_{x2} = M_{x,new}$

Now using previously computed new values, better approximation of area of steel can be done by repeating the same procedure until the required degree of precision is reached. In ESADS, the iteration is completed when M_{xd} *and* $M_{x,new}$ are correct to 6 decimal place in KNm unit.

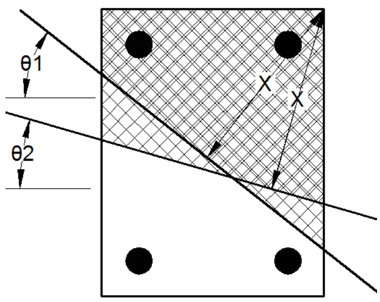
Process-16: Calculating area of reinforcement required to carry the applied load
Input: M_{xd}, M_{yd}, P_d, Material, Geometry
Output: A_s



Flow Chart 12-10 Calculation of area of reinforcement for biaxial column

12.5.4. Angle of Inclination for a Given Moment Ratio and Axial Load

This topic describes how we can calculate the angle of inclination of the neutral axis for a given moment ratio $R = \frac{M_x}{M_y}$ and applied axial load P. Before getting to the procedure we should understand effect of changing the independent variables x and θ on the dependent variables $P, M_x \; and \; M_y$.



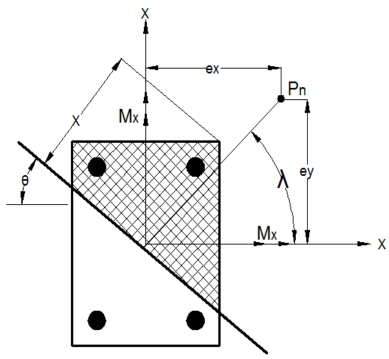
Effect of x and θ on $P, \; M_x \; and \; M_y$

In the design equation derived based on force and moment equilibrium both angle of inclination and depth of neutral axis have contribution. But, the relative effect of x and θ on $P, \; M_x \; and \; M_y$ is different. For example, if we consider the above figure, changing the angle neutral axis have significant effect on the magnitude of $M_x \; and \; M_y$ rather than P. This makes senses, because the area of concrete does not change significantly when the axis is rotated at constant x and the strains of reinforcements in compression and tension zone increase proportionally keeping P relatively unaffected. When you come to the effect on $M_x \; and \; M_y$, changing θ have influence on centroid of the shaded region, which will alter the moments arm and finally change the moments $M_x \; and \; M_y$. But, changing x at constant θ, changes the area of concrete in compression and the relative stress of tension and compression reinforcements. Making P more influenced by changes in x than the moments.

The above principle affects the iteration we use in the design procedure. For example, if we want to change P we can change x keeping θ consant. And if we want to change the magnitude of $M_x \; and \; M_y$ we can change θ keeping x constant. This simplifies the design procedure by limiting the number of independent variables involved in changing one or more dependent variable.

Calculating θ

To get the correct value of x and θ , which will give $P = P_d$ and $\frac{M_x}{M_y} = \frac{M_{xd}}{M_{yd}}$ first we assume initial value of value of $\theta = \tan^{-1}\left(\frac{M_{xd}}{M_{yd}}\right)$



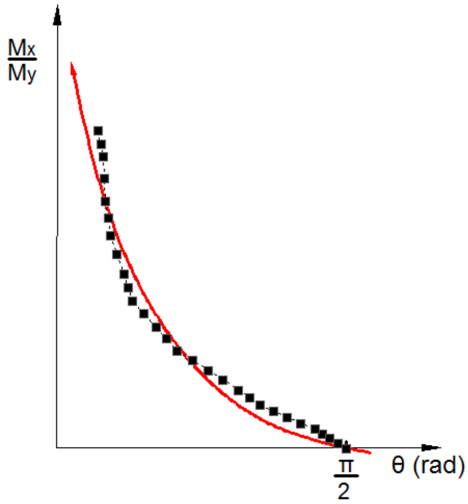
Using this initially assumed value of θ we calculate the depth of neutral axis, which will give $P = P_d$ at the assumed θ as described in Section 12.5.2.

Then, using the calculated value of x we compute $M_x \; and \; M_y$ as summation of force carried by concrete and steel described in Section 4.6.1 and 4.6.2 respectively.

$$M_x = M_{sx}(x, \theta) + M_{sx}(x, \theta)$$

Now calculate the ratio of M_x to M_y. If $\frac{M_x}{M_y} \neq \frac{M_{xd}}{M_{yd}}$ change the value of θ . The value of θ is changed based on the approximate relation of $\frac{M_x}{M_y}$ and θ.

Plotting different values of θ and $\frac{M_x}{M_y}$ at constant x has shown the following relationship between θ and $\frac{M_x}{M_y}$



By regression the cure $y = \frac{c_1}{x} + c_1$ represented by the continuous line in the graph has a better resemblance with the plot of θ and $\frac{M_x}{M_y}$. As we can see from the curve when $= 90° = \frac{\pi}{2}$, $\frac{M_x}{M_y} = 0$, which means the section can carry moment only in x-direction. When $\theta = 0$, $\frac{M_x}{M_y} = \infty$, which means that the section cannot carry moment in y-axis . The following figures show this condition.

To drive a relation between θ and $\frac{M_x}{M_y}$ assume that R $= \frac{M_x}{M_y}$ changes as a function of θ only. Then, we have $R = \frac{c_1}{\theta} + c_1$, where c_1 and c_2 are constants derived from boundary condition. In successive iterations we have two known points $(\frac{\pi}{2}, 0)$ and (θ_1, R_1) from the previous iteration. Substituting this points into the equation $R = \frac{c_1}{\theta} + c_1$ we get c_1 and c_2.

$$c_1 = \theta_1(R_1 + \frac{2\,\theta}{\pi})$$

$$c_2 = \frac{2c_1}{\pi}$$

Finally solving for θ we get,

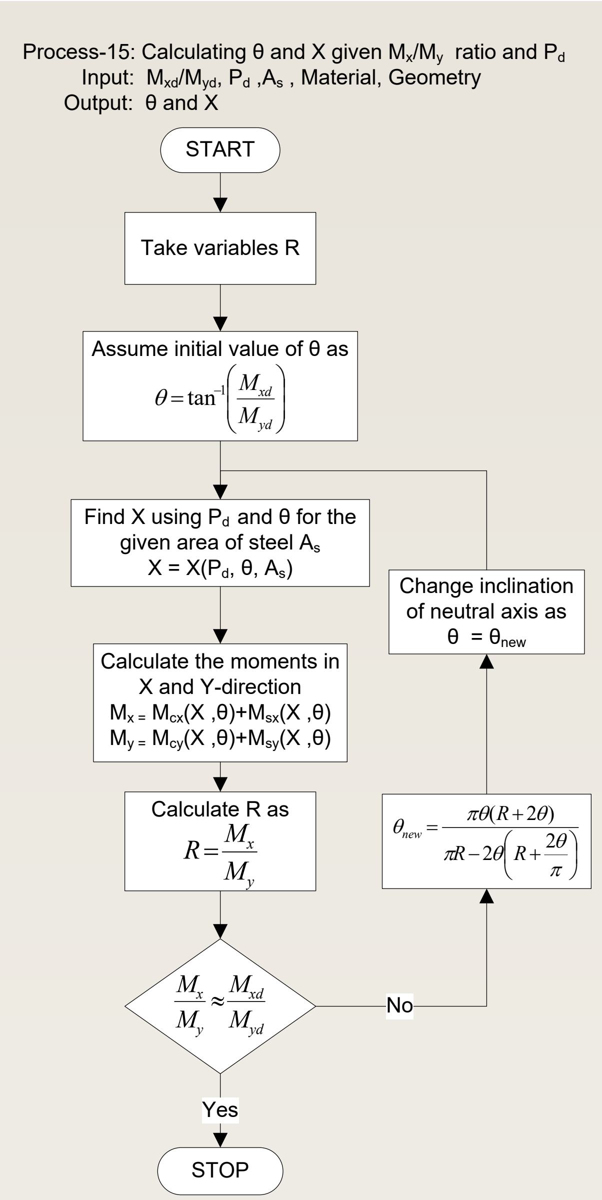
$$\theta_{new} = \frac{\theta_1(R_1 + 2\,\theta_1)}{R_1 - \frac{2\theta_1}{\pi}(R_1 + \frac{2\,\theta_1}{\pi})}$$

Now using the new value of θ_{new} we calculate x required to support an axial load of P_d. Using the calculated x and θ we compute the value of M_x *and* M_y using the description provided in Section 4.6.1 and Section 4.6.2.

$$M_x = M_x(x\,, \theta)$$

$$M_y = M_y(x\,, \theta)$$

If $\frac{M_x}{M_y} \approx \frac{M_{xd}}{M_{yd}}$ end the iteration and the computed values of x and θ are taken as a solution. Otherwise, change the value of θ and repeat the same procedure unit $\frac{M_x}{M_y} \approx \frac{M_{xd}}{M_{yd}}$



Flow Chart 12-11 Calculation of angle of inclination

12.5.5. Depth of Neutral Axis Given Axial Load and Angel of Inclination

To calculate depth of neutral axis x at a given θ and axial load P_d, like uniaxial column first step is calculate initial values of x say x_1 and x_2 which satisfy the following condition.

$$P_1 = N_c(x_1\,,\ \theta) + N_s(x_1\,,\theta) < P$$

$$P_2 = N_c(,x_2\,,\theta) + N_s(x_2\,,\theta) > P$$

After getting x_1 and x_2 the new approximation is made using Regula Falasi Method as,

$$x_{new} = \frac{P-P_1}{P_2-P_1}(x_2 - x_1) + x_1$$

And the corresponding value axial load resistance at $x = x_{new}$, P_{new} is determined as

$$P_{new} = N_c(x_{new}\,,\ \theta) + N_s(x_{new},\ \theta)$$

Now we have to change the value of x_1 and x_2 for the next iteration depending on the newly computed value of x_{new} and P_{new}.

If $P_{new} < P$

$x_1 = x_{new}$

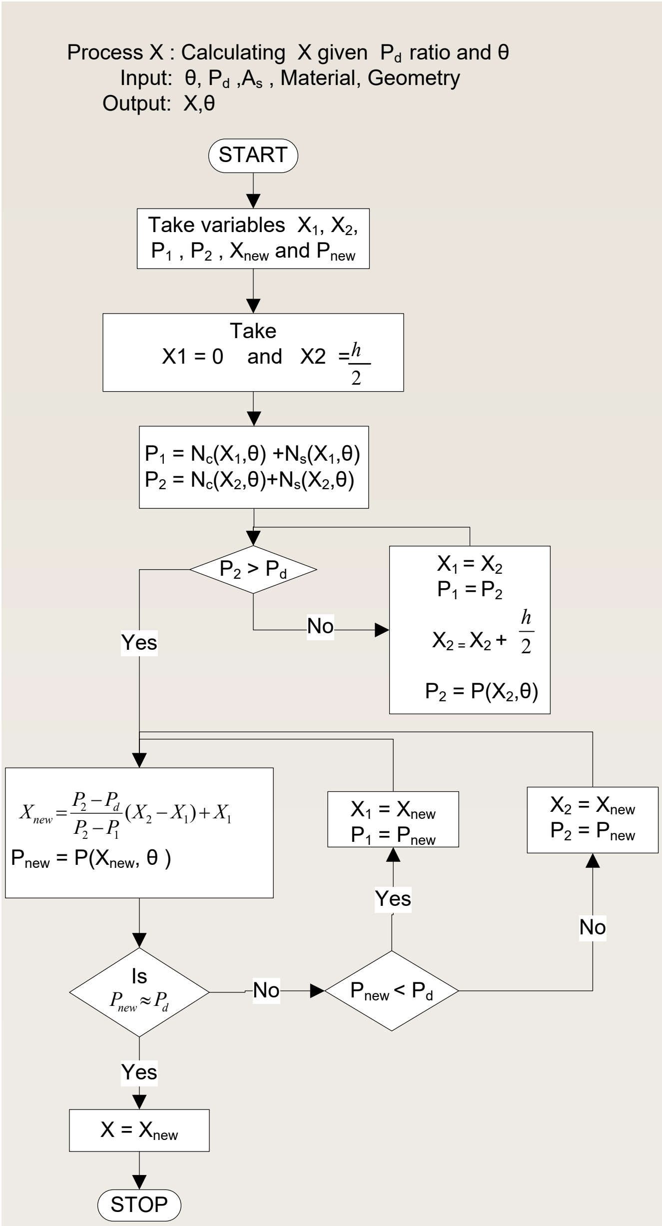
$P_1 = P_{new}$, while $P_2\ and\ x_2$ remain unchanged.

Else if $P_{new} \geq P$

$x_2 = x_{new}$

$P_2 = P_{new}$, while $P_1\ and\ x_1$ remain unchanged.

Finally, using the newly computed value of x_1, x_2, P_1 and P_2 the iteration is proceeded until P_{new} is approximately equal with P_d within a certain degree of precision. See the following figure.



Flow Chart 12-12 Calculation of depth of neutral axis

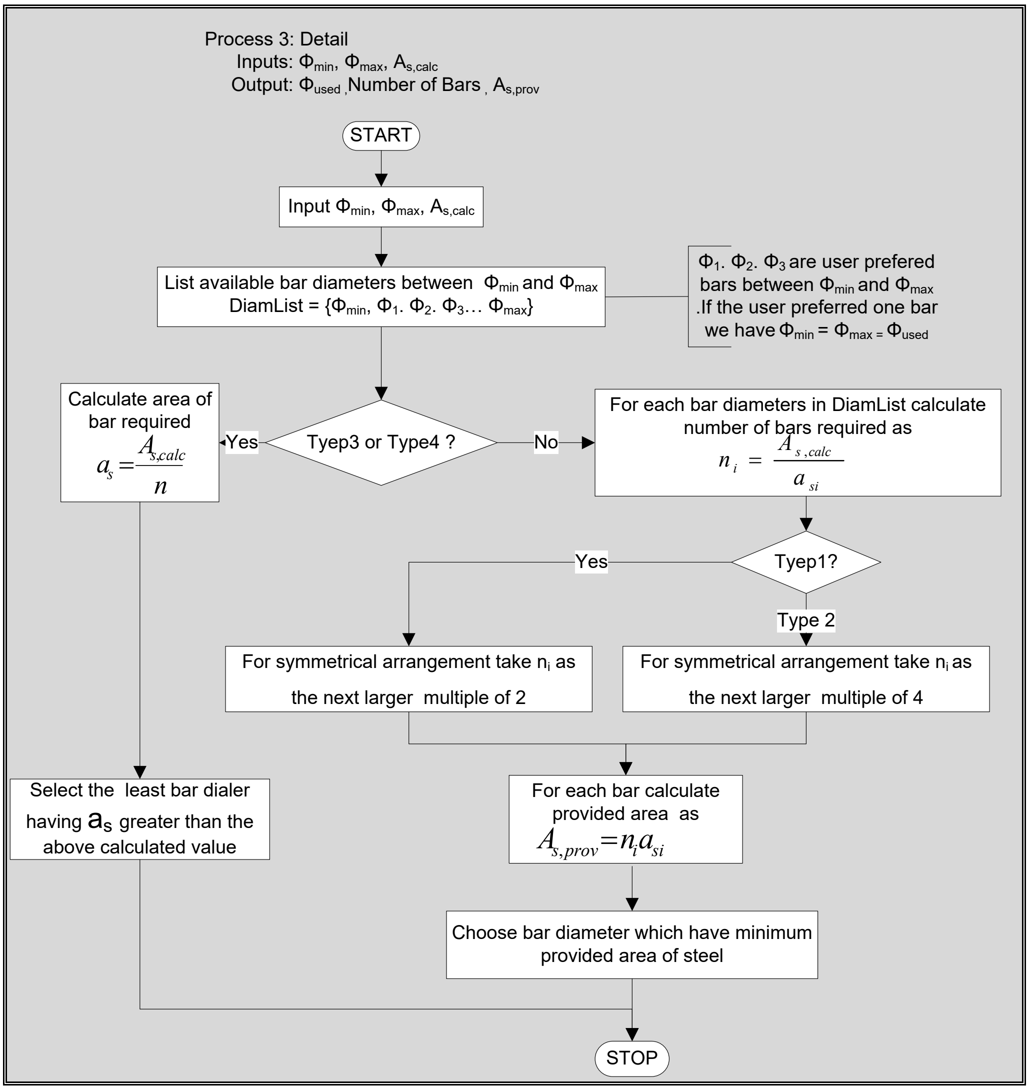
12.6. Detailing

After area of reinforcement calculated for both uniaxial and biaxial columns, we follow the same procedure of detailing. If the user preferred detailing to be done, the software requires the user to input all the necessary data related to detailing. These include,

- ✓ Maximum bar diameter
- ✓ Minimum bar diameter
- ✓ Concrete cover or exposure condition
- ✓ Diameter of tie reinforcement
- ✓ Minimum spacing of tie reinforcing
- ✓ Bar diameter, if the user need for the detailing to be done using only one bar

The first two preferences are optional with the last preference. If the user prefers the detailing to be done using only one bar, there will be no need to input minimum and maximum bar diameter.

If the user preferred detailing to be done, before starting the design the program initially take a certain bar diameter from the limit the user preferred for the detailing. Then, the program computes the location of each reinforcement unit by considering stirrup diameter, concrete cover and main bar diameter. After locating all bars by filing their coordinate, the design is executed and area of reinforcement is calculated. Using the calculated area of steel, a bar between the limit, which will give higher economy and satisfying minimum bar to bar clear spacing requirement is selected. If the selected bar diameter is different from initially assumed bar diameter, the grogram relocate all the reinforcement units using the newly computed bard diameter and execute the design again. This procedure is repeated until initially assumed bar diameter is equal with the selected economical bar diameter. Usually, the iteration converges in few iterations. The fowling flow charts best summaries the detailing procedure followed.



Flow Chart 12-13 Detailing

Process-10: Maximum number of bars per a given width.
Inputs: Φ
Output: Maximum number of bars in x and y-direction.

START

Input Φ

Calculate minimum clear bar spacing
S_{min} = Max{20, Φ_{larg}, Maximum aggregate size+5)}

Calculate maximum number of bar in X-direction

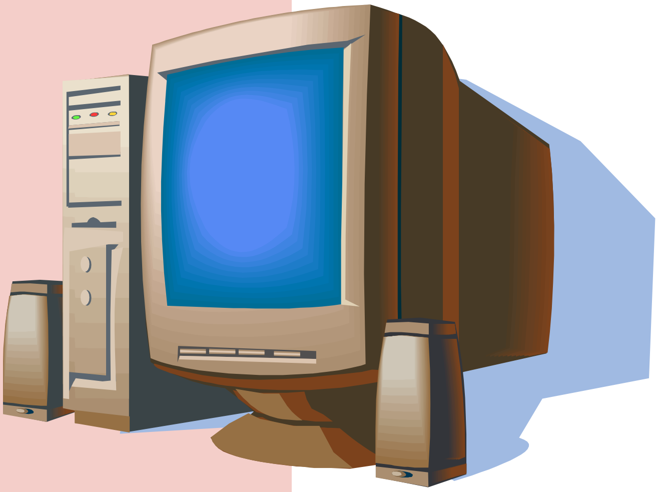
$$\#Xbars = INT\left(\frac{b-2b'+\Phi+S}{\Phi+S}\right)$$

Calculate maximum number of bar in Y-direction

$$\#Xbars = INT\left(\frac{h-2h'+\Phi+S}{\Phi+S}\right)$$

STOP

13. Software Development-Column



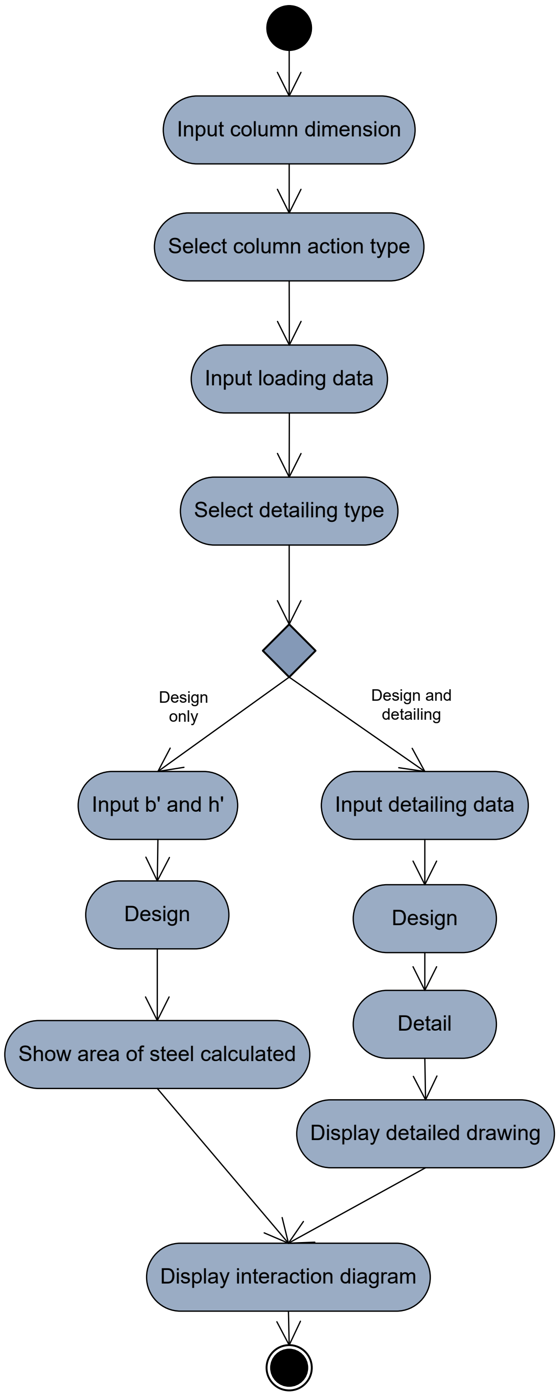
Outlines

- Introduction
- Use Cases
- Object Oriented Modeling

This chapter shortly describes the whole software development procedure used to produce functional columns section design component. The development process starts by preparing Use Cases, which allow scenario based modeling for the software in its graphical user interface (GUI). Then, object oriented modeling is done. Object oriented modeling involves manipulation of operation, expected relation between objects and other object oriented programing issues. In this topic we only describe the most basic issues in software development procedure. Since, the expected readers of this document are Civil Engineering Professionals; more emphasis is given to technical part related to Civil Engineering. But, as a software development project every development procedure and documentation should be described. Therefore, the need of this chapter would be more critical for the sack of completeness only. Software Engineering extremely wide, describing every steps and procedures of development take long time more effort. Considering, this only very important topics are covered here.

13.1. Use Cases

Use cases are scenario based softwares modeling techniques, which describe the interaction between the user and the software. In addition, use cases better characterize requirements and build meaningful analysis and design model. Herein we will describe all the possible action that the user can do, while interaction with software. These actions can be described word by word to give narrative use cases or represented diagrammatically using activity diagram. For ease of understanding and graphical presentation we prefer to present the use case diagrammatically using activity diagrams.



13.2. Object Oriented Modeling

Object oriented modeling describes objects that the software will manipulate, the operations (methods) involved in them, attributes contained, interaction with other objects and relationship between objects. The relationship can be association or inheritance. The first step in object oriented modeling is to identify the possible objects, which can be represented by classes. Here we only discuss those objects specific to this column design component. In the modeling process first we have identified all the necessary objects involved. Then, we specify object attributes, define operations (Methods), and express associations and relationships (Inheritance).

13.2.1. Identification of Classes

Classes are encapsulated in name spaces and name spaces are encapsulate in assemblies. As it is described in the first part of this paper our assemblies are classified based on their functionality and place of application. The overall system for column design components mainly involves two assemblies.

- ✓ ESADS.Mechanics.Design.Column.dll
- ✓ ESAS.Graphics.Column.dll

Objects in ESADS.Mechanics.Design.Column.dll assembly

Object	Name	Type
Column	eColumn	Abstract Class
Uniaxial column	eUniaxial	Class
Biaxial Column	eBiaxial	Class
Reinforcement Unit	eAs	Structure
Concreter Section	eConc	Structure

Column: abstract class which defines basic methods and attributes common to all biaxial and uniaxial columns. Instance of this class cannot be created.

Uniaxial Column: class derived from the base class column and defines those attributes and method unique to uniaxial column only. Instance of this class can be created by passing the basic parameters through the constructor.

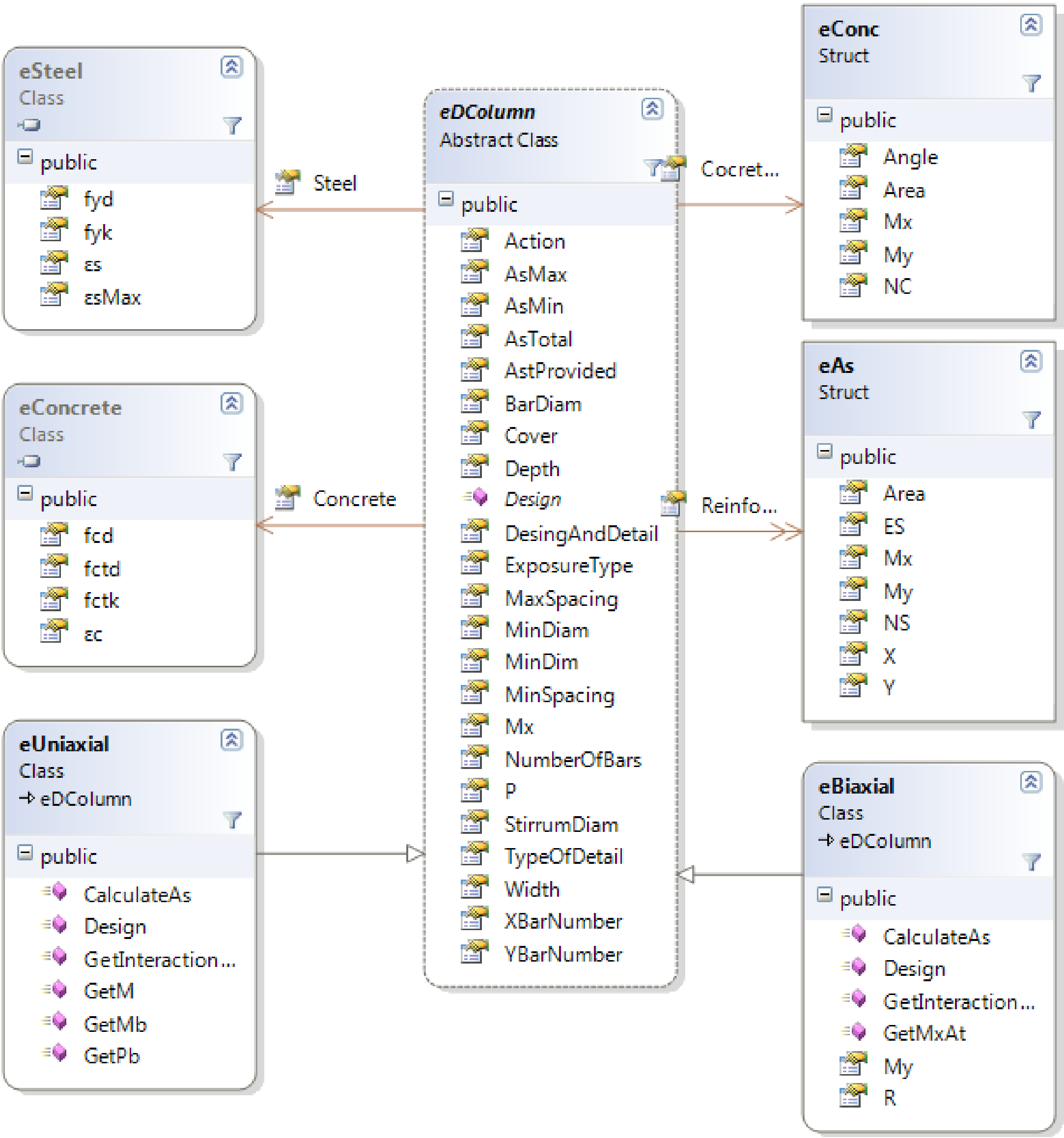
Biaxial Column: Class derived from base class Column and defines all methods and attributes unique to it. These classes contain all the necessary methods, which are used to design biaxial column.

Reinforcement Unit: A structure, which contains all the necessary methods and attributes used for reinforcement units lie coordinate strain stress level and area.

Concrete Section: A structure which contains all the necessary methods /processes and attributes used for concrete section for a column. The attributes include moment capacity of the concrete section, axial load carried, depth of neutral axis n others.

13.2.2. Class Diagram

The following diagram show relation, association and aggregation of all objects involved in ESADS.Mechanics.Design.Column.dll assembly.



eSteel
Class
public
fyd
fyk
εs
εsMax
Steel
eDColumn
Abstract Class
public
Action
AsMax
AsMin
AsTotal
AstProvided
BarDiam
Cover
Depth
Design
DesingAndDetail
ExposureType
MaxSpacing
MinDiam
MinDim
MinSpacing
Mx
NumberOfBars
P
StirrumDiam
TypeOfDetail
Width
XBarNumber
YBarNumber
Cocret...
eConc
Struct
public
Angle
Area
Mx
My
NC
eConcrete
Class
public
fcd
fctd
fctk
εc
Concrete
Reinfo...
eAs
Struct
public
Area
ES
Mx
My
NS
X
Y
eUniaxial
Class
eDColumn
public
CalculateAs
Design
GetInteraction...
GetM
GetMb
GetPb
eBiaxial
Class
eDColumn
public
CalculateAs
Design
GetInteraction...
GetMxAt
My
R

14. Checking and Illustration

Let's check it!!

Outline:

- ❑ Example 1: Uniaxial Column Type1
- ❑ Example 2: Uniaxial Column Type3

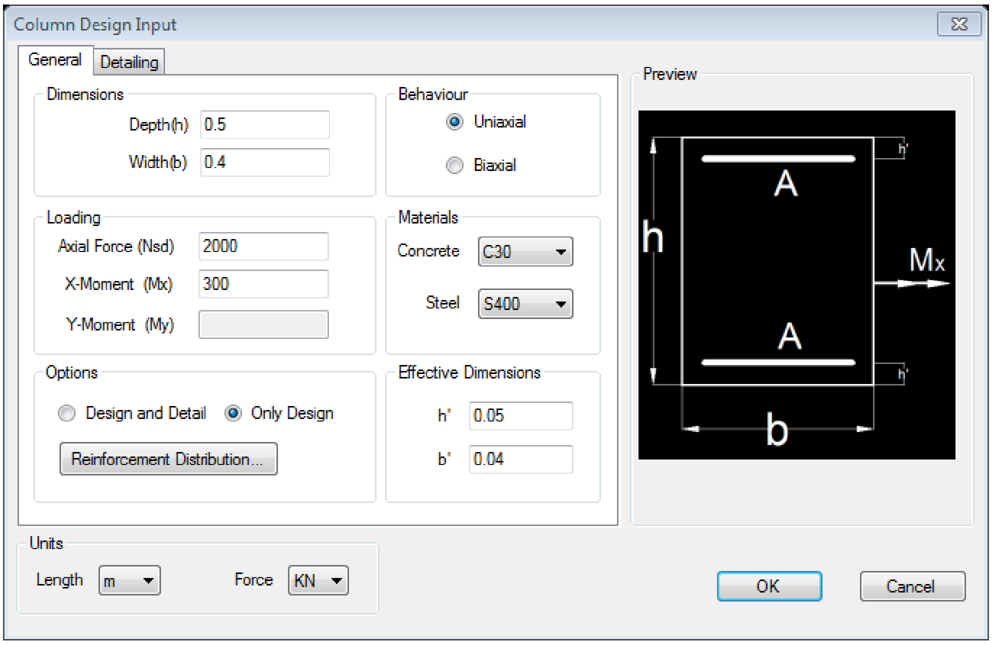
In this chapter the software output are check and illustrative examples are done using the software. Checking the reliability of the software out can be done using excel sheet or manually depending on the bulkiness of the calculations. As an option, the result from the software can be checked using design charts provided in EBCS2 Part-2. Based on the generated output, some recommendation about the result is also given here. Therefore, by doing illustrative example and simultaneously performing the check we will grants the reliability of the software output. In the checking process only basic outputs, which are expected to be necessary are presented.

For uniaxial column only two examples are done using detailing Type1 and detailing Type3. Then, the result taken from the software is checked using excel sheet. Similarly, the output will be also checked using the design chars provide in in EBCS-2 PART-2 for uniaxial column. Since addressing all the process and outputs is difficult, here only main processes and outputs are provide.

Example1: Uniaxial Column Type1

1 Input general data related to the design these may include.

- ✓ Geometry
- ✓ Material
- ✓ Action/ Behaviour
- ✓ Loading

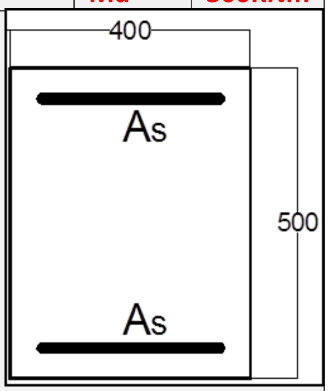


2 Run the design and detailing.

Trial 1:

Using As = $A_{s,min}$ = 0.008bh/2 = 800mm² and solving x which give P = Pd = 2000KN, we get X = 410.5mm

b	**400**		**Pd**	**2000KN**
h	**500**		**Md**	**300KNm**
d'/h	**0.1**			
d'	**50**			
As	**800**			
As'	**800**			
fcd	**13.6**			
fyd	**347.826**			
Eyd	**0.00174**			
Ecu	**0.0035**			
d	**450**			
E	**200000**			



X	**Nc**	**Ns1**	**Ns2**	**P**	**Mc**	**Ms1**	**Ms2**	**M**
410.5	**1775.623**	**278.26**	**53.883**	**2000**	**152.347**	**55.652**	**10.777**	**218.78**

Trial 2:

Now the software estimated the second area of steel to be As = 1505.24mm². Then, solving for X which will give P = 2000KN we get. X = 384.9047mm

X	Nc	Ns1	Ns2	**P**	Mc	Ms1	Ms2	**M**
384.9047	1654.634	523.56	178.2	**2000**	158.908	104.71	35.639	**299.26**

Trial 2:

Third estimate of by interpolation of the previous two values gives As = 1511.01mm². Then, solving for X for Pd gives X = 384.74mm

X	Nc	Ns1	Ns2	**P**	Mc	Ms1	Ms2	**M**
384.7403	1653.84	525.57	179.41	**2000**	158.94	105.11	35.882	**299.94**

Now as it can be seen, from the last two iterations converged, the depth of neutral axis and area of steel is not changing significantly. Therefore we can end up the iteration. Taking the design out puts as.

As = 1505.24mm^2

P_{max} = 3730.4KN

P_b(balanced) = 1287.75KN

M_b(balanced) = 377.39KNm

M(pure bending) = 216.6KNm

Checking using design charts:

$$\mu = \frac{M_u}{A_c f_{cd} h} = 0.221$$

$$v = \frac{N_u}{A_c f_{cd}} = 0.735$$

From chart No.2(h'/h = 0.1) we get ω = 0.4

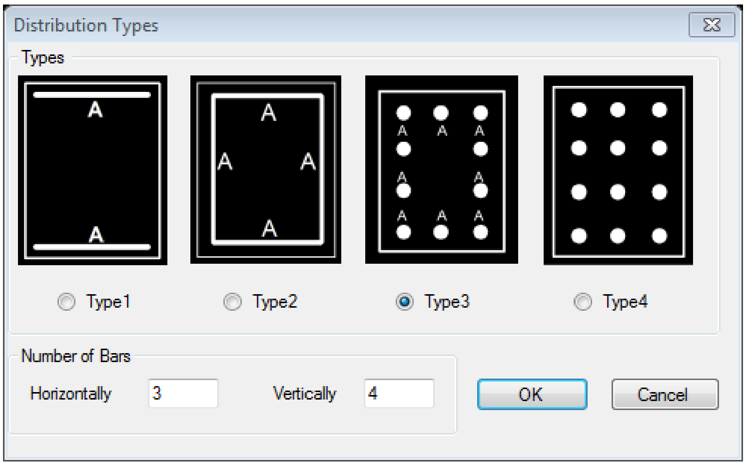
$$A_{s,total} = \frac{\omega A_c f_{cd}}{f_{yd}} = 3128.00$$

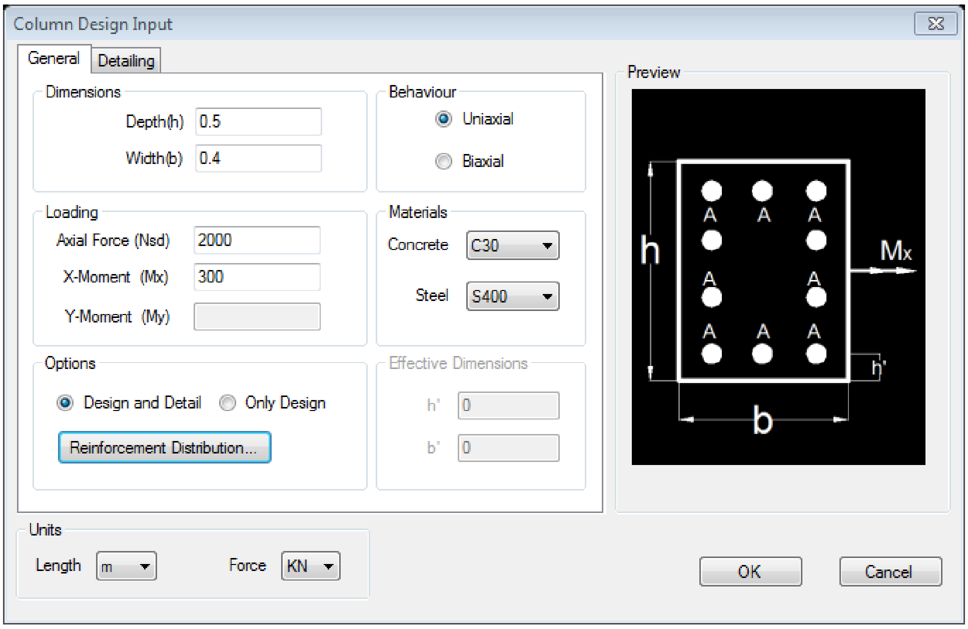
As = 3128.00/2 = 1564.00mm^2 >1511.01mm^2 . As we can see, the code value are somewhat conservative.

Example2: Uniaxial Column Type3

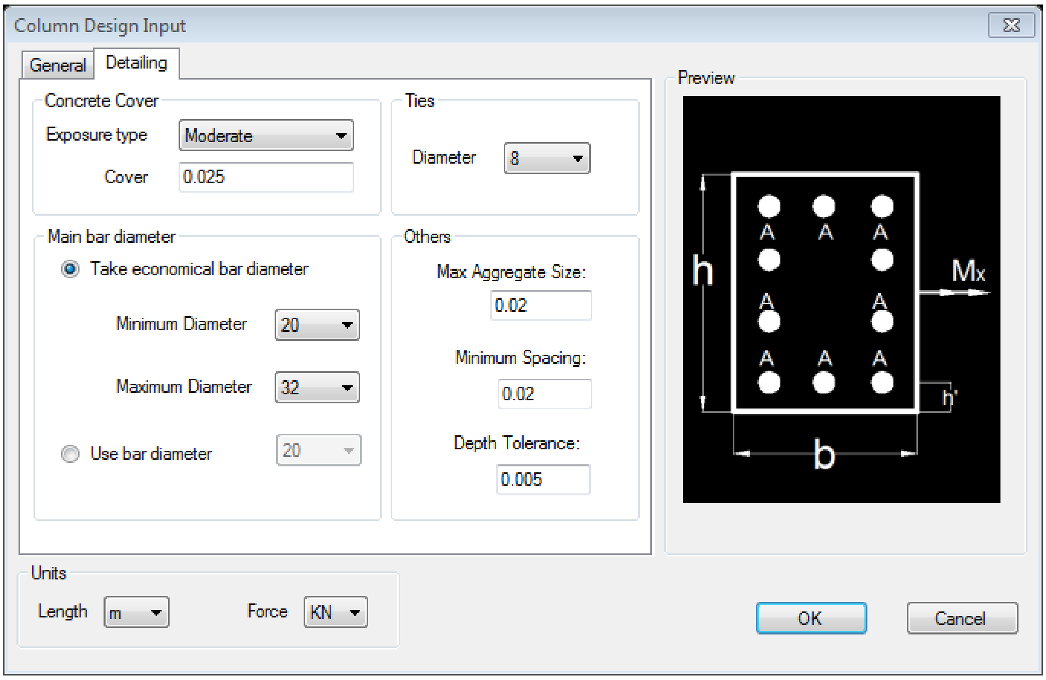
1 Input general data related to the design these may include.

- ✓ Geometry
- ✓ Material
- ✓ Action/ Behaviour
- ✓ Loading
- ✓ Select detailing type.





2 Input detailing data.

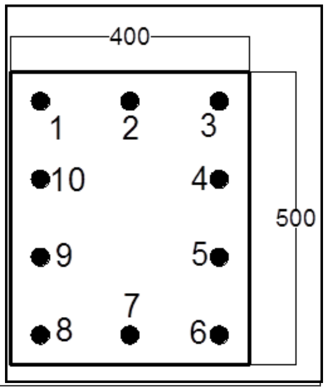


3 Run the design and detailing.

Trial 1

When the design is executed the program in initial bar diameter and start the design using diameter 20 bar we have h' = 20/2+8+25 = 43. And assumes initial area of steel of each bar A = A_{min}/10 = 160mm². Solving for X , which gives P = Pd =2000KN the program get X = 399.773mm using this value of X we calculate the force and moment carried by each steel as following. The calculation is done on excel sheet.

b	400		Pd	2000KN
h	500		Md	300KNm
d'/h	0.1			
d'	43			
As	1511.01			
As'	1511.01			
fcd	13.6			
fyd	347.826			
Eyd	0.00174			
Ecu	0.0035			
d	457			
E	200000			



Bar	Area		Z	εs	fs	N	M
1	160	207	356.7727	0.0031235	347.8261	55.652	11.52
2	160	207	356.7727	0.0031235	347.8261	55.652	11.52
3	160	207	356.7727	0.0031235	347.8261	55.652	11.52
4	160	69	218.7727	0.0019153	347.8261	55.652	3.84
5	160	-69	80.77273	0.0007072	141.4326	22.629	-1.561
6	160	-207	-57.2273	-0.000501	-100.2047	-16.033	3.3188
7	160	-207	-57.2273	-0.000501	-100.2047	-16.033	3.3188
8	160	-207	-57.2273	-0.000501	-100.2047	-16.033	3.3188
9	160	-69	80.77273	0.0007072	141.4326	22.629	-1.561
10	160	69	218.7727	0.0019153	347.8261	55.652	3.84
Sum of force and moment carried by steel						275.42	49.074
Force and Moment carried by Concrete						1724.6	155.37
CONCRETE+STEEL						***2000***	***204.44***

Trial2:

Now since the moment carried (204.44KN) is less than M_d =300KN at the same axial load the area of steel should be increase. Finally, the software calculate the second estimate of area of steel A = 353.mm^2 and X = 359.57mm.

Bar	Area	Y	Z	εs	fs	N(KN)	M(KNm)
1	353.013	207	316.5775	0.0030815	347.8261	122.79	25.417
2	353.013	207	316.5775	0.0030815	347.8261	122.79	25.417
3	353.013	207	316.5775	0.0030815	347.8261	122.79	25.417
4	353.013	69	178.5775	0.0017382	347.6421	122.72	8.46784
5	353.013	-69	40.57755	0.000395	78.99348	27.886	-1.9241
6	353.013	-207	-97.4225	-0.0009483	-189.6551	-66.951	13.8588
7	353.013	-207	-97.4225	-0.0009483	-189.6551	-66.951	13.8588
8	353.013	-207	-97.4225	-0.0009483	-189.6551	-66.951	13.8588
9	353.013	-69	40.57755	0.000395	78.99348	27.886	-1.9241
10	353.013	69	178.5775	0.0017382	347.6421	122.72	8.46784
Sum of force and moment carried by steel						**468.73**	**130.915**
Force and Moment carried by Concrete						**1531.3**	**162.574**
CONCRETE+STEEL						***2000***	***293.489***

Trail3:

After interpolation, A = 366.02mm^2 and X = 357.81mm.

Bar	Area	Y	Z	εs	fs	N(KN)	M(KNm)
1	366.028	207	314.8096	0.0030794	347.8261	127.31	26.354
2	366.028	207	314.8096	0.0030794	347.8261	127.31	26.354
3	366.028	207	314.8096	0.0030794	347.8261	127.31	26.354
4	366.028	69	176.8096	0.0017295	345.9011	126.61	8.73606
5	366.028	-69	38.80959	0.0003796	75.92506	27.791	-1.9176
6	366.028	-207	-99.1904	-0.0009703	-194.0509	-71.028	14.7028
7	366.028	-207	-99.1904	-0.0009703	-194.0509	-71.028	14.7028
8	366.028	-207	-99.1904	-0.0009703	-194.0509	-71.028	14.7028
9	366.028	-69	38.80959	0.0003796	75.92506	27.791	-1.9176
10	366.028	69	176.8096	0.0017295	345.9011	126.61	8.73606
Sum of force and moment carried by steel						**477.66**	**136.808**
Force and Moment carried by Concrete						**1522.3**	**162.702**
CONCRETE+STEEL						***2000***	***299.51***

Trail4:

Finally on the last trail we get $A = 367.09mm^2$ and $X = 357.67mm$.

Bar	Area	Y	Z	ε_s	fs	N(KN)	M(KNm)
1	367.087	207	314.6691	0.0030792	347.8261	127.68	26.4302
2	367.087	207	314.6691	0.0030792	347.8261	127.68	26.4302
3	367.087	207	314.6691	0.0030792	347.8261	127.68	26.4302
4	367.087	69	176.6691	0.0017288	345.7619	126.92	8.7578
5	367.087	-69	38.66907	0.0003784	75.67988	27.781	-1.9169
6	367.087	-207	-99.3309	-0.000972	-194.4022	-71.362	14.772
7	367.087	-207	-99.3309	-0.000972	-194.4022	-71.362	14.772
8	367.087	-207	-99.3309	-0.000972	-194.4022	-71.362	14.772
9	367.087	-69	38.66907	0.0003784	75.67988	27.781	-1.9169
10	367.087	69	176.6691	0.0017288	345.7619	126.92	8.7578
Sum of force and moment carried by steel						**478.37**	**137.289**
Force and Moment carried by Concrete						**1521.6**	**162.711**
CONCRETE+STEEL						***2000***	***300***

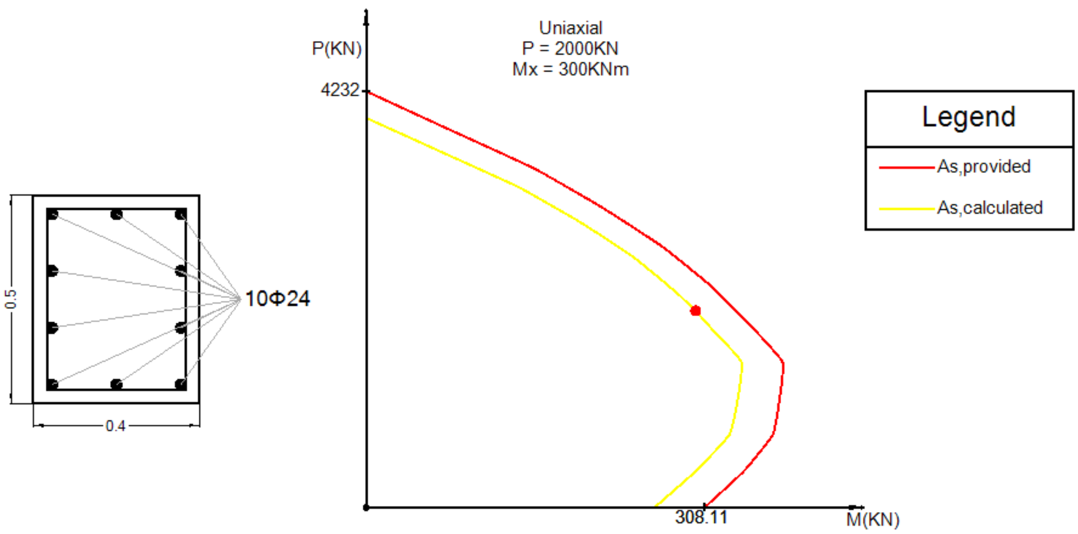
As we can see from the table the iteration converged. The next step is to detail the section. Using $A = 367.09$, the least bar which have area above the calculated value is Φ_{20} with $452.39mm^2$ area. But if we used this diameter h' will be chanced and the whole design should be revised using the newly calculated diameter. Now we have h' = 25+8+24/2 = 45. Since the h' is increased, using then new value of h' we revise the design. After refilling the coordinate of each bar and taking previously calculated area of steel $A_s = 367.087mm^2$, the program calculate the new X to be 357.20mm.

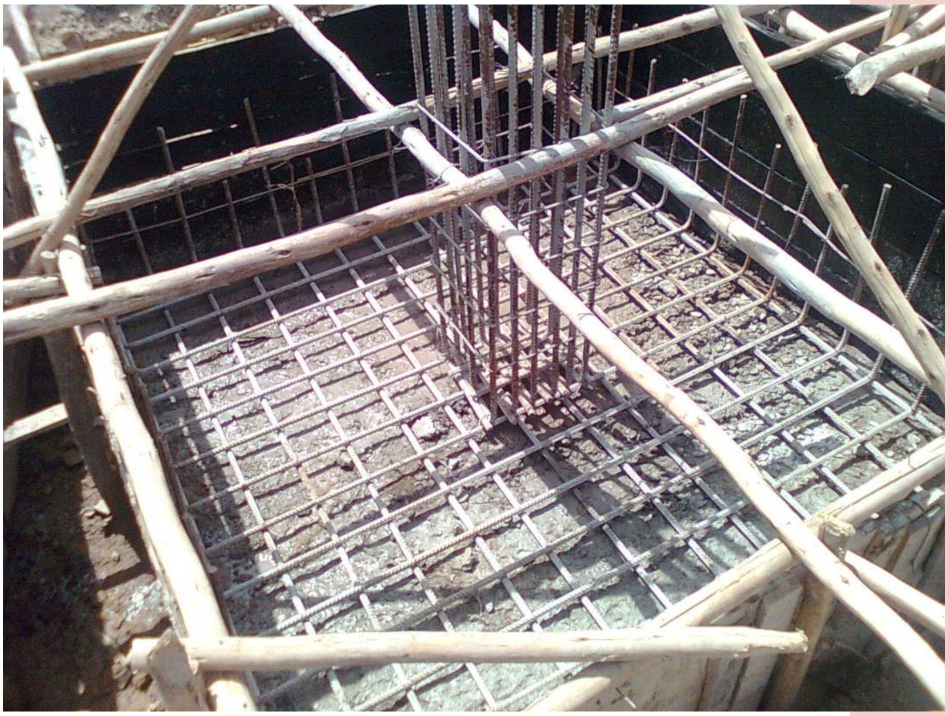
Bar	Area	Y	Z	ε_s	fs	N(KN)	M(KNm)
1	367.087	205	312.1958	0.0030591	347.8261	127.68	26.1749
2	367.087	205	312.1958	0.0030591	347.8261	127.68	26.1749
3	367.087	205	312.1958	0.0030591	347.8261	127.68	26.1749
4	367.087	68.3333	175.5292	0.0017199	343.9861	126.27	8.62864
5	367.087	-68.333	38.8625	0.0003808	76.15921	27.957	-1.9104
6	367.087	-205	-97.8042	-0.0009583	-191.6677	-70.359	14.4235
7	367.087	-205	-97.8042	-0.0009583	-191.6677	-70.359	14.4235
8	367.087	-205	-97.8042	-0.0009583	-191.6677	-70.359	14.4235
9	367.087	-68.333	38.8625	0.0003808	76.15921	27.957	-1.9104
10	367.087	68.3333	175.5292	0.0017199	343.9861	126.27	8.62864
Sum of force and moment carried by steel						**480.43**	**135.232**
Force and Moment carried by Concrete						**1519.6**	**162.779**
CONCRETE+STEEL						***2000***	***298.01***

Finally after two iteration the program calculated A = 371.43mm^2 and X = 356.62mm. Then the final output is summarized as.

Bar	Area	Y	Z	ε_s	fs	N(KN)	M(KNm)
1	371.437	205	311.6207	0.0030584	347.8261	129.2	26.4851
2	371.437	205	311.6207	0.0030584	347.8261	129.2	26.4851
3	371.437	205	311.6207	0.0030584	347.8261	129.2	26.4851
4	371.437	68.3333	174.954	0.0017171	343.412	127.56	8.71632
5	371.437	-68.333	38.28735	0.0003758	75.15309	27.915	-1.9075
6	371.437	-205	-98.3793	-0.0009655	-193.1058	-71.727	14.704
7	371.437	-205	-98.3793	-0.0009655	-193.1058	-71.727	14.704
8	371.437	-205	-98.3793	-0.0009655	-193.1058	-71.727	14.704
9	371.437	-68.333	38.28735	0.0003758	75.15309	27.915	-1.9075
10	371.437	68.3333	174.954	0.0017171	343.412	127.56	8.71632
Sum of force and moment carried by steel						**483.35**	**137.185**
Force and Moment carried by Concrete						**1516.7**	**162.815**
CONCRETE+STEEL						***2000***	***300***

Therefore, we use 10Φ24 bars and the program shows the detailed drawing and interaction diagram for provided and calculated area of steel. The red line in the interaction diagram is drawn based on the provided area of steel and the yellow line is drawn using the calculated area of steel. The red dot indicates the location of P_d and M_d in the interaction diagram.





Reinforcement cage of footing ready to be casted at Adama for G+9 building of Hawas Agri-Business
Courtesy:
The reinforcements are bent at the end in alternating pattern to save overall reinforcement economy.

PART IV

ISOLATED FOOTING

Chapters:

- ❑ Introduction to Footing
- ❑ Existing Design Practices – Footing
- ❑ Structural Design of Footing
- ❑ Development – Footing

ESADS V 1.0

15. Introduction to Footing

Outline:

- Overview
- Scope
- Features

In this chapter, we are going to discuss the basic features of the footing component.

15.1. Overview

Footing is an artificially built part of a building which is used to safely transfer the loads coming from super structure of the building to the soil. In order to fulfill its safe transferring purpose it should be carefully designed. Designing a footing just by hand may be possible but the procedure we should follow is tedious and time consuming. Because of these reasons there may be a consequence of faulty design which leads to lots of destruction after the footing is constructed. As an engineer, to prevent the happening of such problems, we need to find a solution which avoids those destructions during construction and after the building starts giving its service. That's why designing an isolated footing by considering EBCS has become one of the components of ESADS. This software is different from other footing design software is that it considers EBCS code. Thus, this software is helpful in getting the design output in the most safe and understandable way.

This part concerns about issues related to scope and features, existing design practice, design procedure, software development and there is also worked example to briefly explain the whole working system of the software. Each of these topics is explained briefly in the following subsequent chapters. In addition, in this chapter scope and features of ESADS related to footing are discussed. Under scope part things we have done and we are going to do in near future are described. The best aspects while designing a footing are explained in feature part of this chapter.

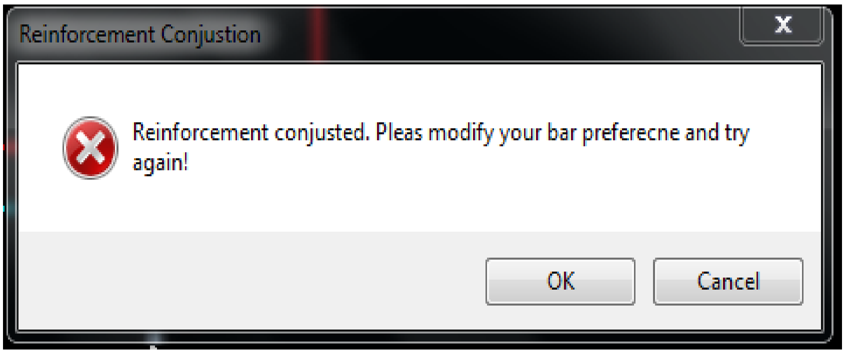
15.2. Scope

As the time given to complete the project is not sufficient enough to cover all what we have intended to do on the scope of footings, we have been obligated to restrict our component to be designed. Considering the scope of application, and the urgency of getting design aids on the area, we have decided to develop software which designs an isolated footing capable of supporting concrete column. The range of application of such a footing is still wide. There are footings that support pedestal, wall, and concrete column with steel base plate; yet they are not included in this first edition. The operation of the software can be implemented on rectangular and square footing shapes with the exception circular. The design procedure is also doesn't follow soil proportioning method and it considers only footing rested on soil rather than footing rested on pile. Though this software seems to have shortcomings, it is an ice breaking work on the scope of footing designs based on EBCS. It opens a way for better improvements to come in the near future. Since there is so much more to be done on the sector we strongly hope to provide the better version of the software which can address all the types of footings with specific characteristics.

15.3. Features

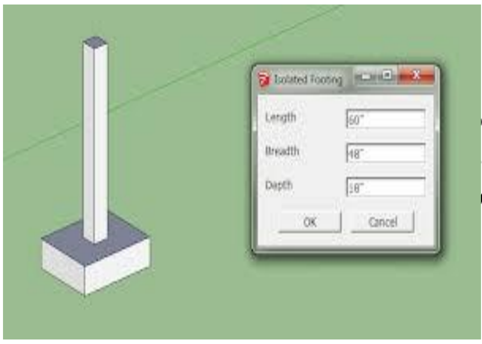
As any software our software has also inbuilt feature which let the user to perform design. Describing all the features may be unimportant. But here we describe the most general ones which make out software different from other footing structural design softwares. Generally, the software has different features some of these are:

- ✓ The output displays with detailed information such as section drawing, number of bar, diameter, length and their arrangement
- ✓ The user is more flexible to insert his bar and spacing preferences.
- ✓ If the data inserted by the user are failed after designing, the software can suggest him what the problem is and how to take a measurement for the failed design. In this feature for example, if the bar diameter selected by the user makes the reinforcement arrangement congested, the system displays a dialog box which tell the user to modify his input in the following way



- ✓ After the user inserted the data which are required to start the design process, he can check geometry of the footing and footing column and loading before running the design.
- ✓ The design output displays immediately after the user clicks on “Run Design” from the run menu or just after he clicks on “F6”

16. Existing Design Practices – Footing



Chapter Outline:

ftware

mplates

lethod

In this chapter, we are going to show description on how existing design practice works and their limitations. Since there is no another software which designs a structural component considering EBCS most of Ethiopian Engineers are forced to use other design aids. But this way is an approximation method and it is not fully safe to apply on the construction area. The limitations of such design aids are one of the reasons to initiate us to develop the software which can design the component we need considering EBCS. There are three ways of designing a footing those are listed below

- ✓ Design software
- ✓ Design Templates
- ✓ Manual Method

16.1. Design software

There are different design softwares which are capable of designing footing. From those softwares SAFE is one of them. It is an ultimate integrated tool for designing reinforced and post tensioned concrete and foundation system by considering around 10 codes. Some of them are ACI 318-8, AS 3600-2009, Euro code 2-2004 and Hong Kong CP-2004.

16.1.1. Limitation

- ✓ It doesn't consider EBCS requirements.
- ✓ It doesn't show the preview of the geometric properties of the footing and its column and the given loading before the user runs the design.
- ✓ It doesn't find the thickness of the footing. So, the user is expected to insert the depth.
- ✓ Its interaction with the user is somehow complicated if the user is not expert on it

- ✓ The user may get confused about the design output.
- ✓ In the Run menu Run analysis and Run design are found together and it is not good practice to make the user more flexible.

16.2. Footing Design Templates

It is another way of designing the footing using excel sheet as its working environment. When the user inserts different parameters of the footing the calculation is done automatically using the given data.

16.2.1. Limitation

- ✓ The user cannot adjust the units by himself. So, the user is expected to change the units manually and inset to the template.
- ✓ Here also the user may get confused of where did some calculated parameters come. For example the final values of k1 and K2 are displayed without their formula. It makes hard to check whether their values are correct or not
- ✓ Detailing of the design is not part of the template and there is no diagram which shows the last reinforcement arrangement of the bars. Therefore the user is forced to use manual method or other design software to complete the full design output.

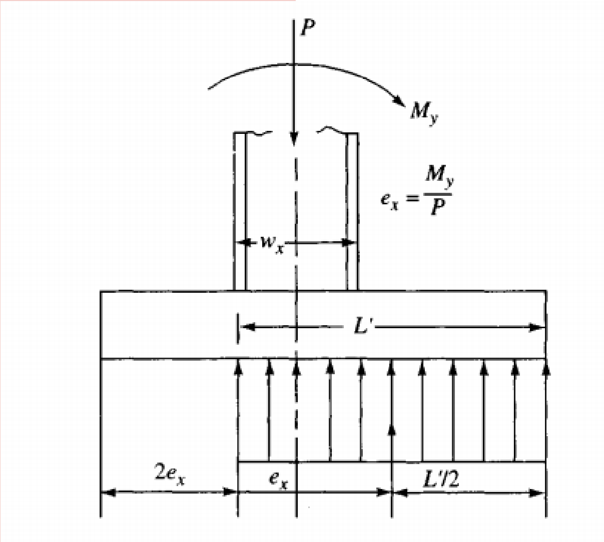
16.3. Manual Method

This is a method of designing a footing which is the designer himself do the design by following the design procedures. In order not to use the software which doesn't consider EBCS the user is forced to use this method of designing.

16.3.1. Limitation

- ✓ As the procedure is long and tedious it may leads to an erroneous design.
- ✓ Revisions till the safest and economical value of depth and diameter is known is difficult here. Because of its tediousness the user can take the value which is safe but near to be economical.
- ✓ The designer should be very careful while calculating different values searching the wrong value will take additional time.

17. Structural Design of Footing



Outlines:

- General Procedure
- Detailed procedure
- Flow chars

In this chapter the procedures that the software follows to reach the design outputs are discussed. We have covered the entire design procedure of the software in a clear manner. There are also flowcharts which show the flow of the designing procedure diagrammatically. The steps which are listed below are the easiest way to show how it works for those people who are not familiar with code which is written by C# language and generally by any programming languages. As the software is capable of designing footing with eccentric loading and without eccentric loading, the steps listed below can be for both cases depending on the data inserted by the user. The procedures goes through specific calculations and each of them are explained below

The procedures are showed in four ways

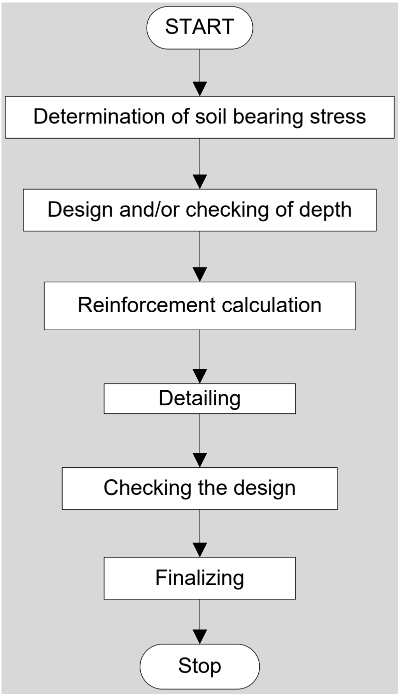
- ✓ General Procedure
- ✓ Detailed procedure
- ✓ Flow chart of general procedure
- ✓ Flow chart of detailed procedure

17.1. General Procedures

The structural design of footing fist starts by calculating the resisting stress under the footing. Then, depending on whether the user provided depth or not we check or calculated the depth required considering punching and wide beam shear. Finally, reinforcements are calculated using the maximum moment developed in the footing and detail is done for the calculated reinforcement. The following seven steps shortly summarize the whole design procedure of reinforced concrete footing.

1. Determination of soil bearing stress

2. Checking the adequacy of the depth for wide beam shear and punching shear if the depth is given
3. Designing the depth which is safe for wide beam shear and punching shear if the depth is not given
4. Reinforcement calculations
5. Detailing
6. Checking the Design
7. Finalizing



17.2. Determination of Soil Bearing Stress

The first procedure while designing an isolated footing is that calculation of ultimate and average bearing stress which are used in punching shear and wide beam shear computation. When footings have overturning moments as well as axial loads, the resultant soil pressure does not coincide with the centroid of the footing. The center of the resultant uniform soil pressure is at the centroid of B' L' rectangle and is also at the eccentric distance(s) e_L and/or e_B which are computed from the column centre. In the dialog box, the user can specify the footing if it has an eccentric loading by giving the value for moments and depending on that the soil bearing stresses are calculated as follows:

$$q_u = \frac{P}{L'B'} \qquad L' = L - 2e_L \qquad e_L = \frac{M_B}{P}$$

$$q_{avg} = \frac{P}{L*B} \qquad B' = B - 2e_B \qquad e_B = \frac{M_L}{P}$$

If the footing is not eccentrically loaded the resultant uniform soil pressure is at the centroid of BL center. Therefore, the value of the ultimate stress and average stress become equal.

$$q_u = q_{avg} = \frac{P}{L*B}$$

17.3. Check for Wide Beam Shear

Depending on the footing column type selected by the user, wide beam shear and punching shear should be checked in both directions. We should check the wide beam shear primarily at d distance from the face of the critical column and then the punching shear at 1.5d distance. EBCS requires the wide beam shear resistance of concrete shall be taken as

$$V_{rd} = 0.25 f_{ctd} K_1 K_2 b_w d$$

If the user selects rectangular footing column type in the dialog box, the checking for wide beam shear will be in such a way that the acting wide beam shear is calculated and be checked with the resistance of the concrete V_{rd}.If acting wide beam shear, V_w ,is greater than $V_{rd}$the depth should be revised till it fulfill the requirement. In the longest direction of the footing V_w is computed as

$$V_w = \left({}^{B}/_{2} - {}^{b_c}/_{2} - deffB\right) * L * q_{ult}$$

Also in the shortest direction of the footing V_w is computed as

$$V_w = \left({}^{L}/_{2} - {}^{l_c}/_{2} - deffL\right) * B * q_{ult}$$

If the user selects circular footing column type instead of rectangular, the checking procedure is the same but the only difference is when calculating acting wide beam shear and it is simply found by replacing column diameter in terms of b_c and l_c in the longest and shortest direction respectively.

In the longest direction V_w =(B⁄2-c_diam⁄2-d_{effB})$* L * q_{ult}$

In the shortest direction $V_w = \left({}^{L}/_{2} - {}^{c_{diam}}/_{2} - d_{effL}\right) * B * q_{ult}$

17.4. Check for Punching Shear

After checking the adequacy of the depth for wide beam shear and if it is satisfied the next procedure is to check that depth for punching shear is adequate or not. The punching shear generally controls the depth of the footing. There is no need of checking the depth for punching shear in both directions. Because it is depend on the perimeter of punching. But we should consider which footing column type is selected by the user (circular or rectangular). The critical section that is to be considered is 1.5d distance from the face of footing column. As EBCS requires the resistance of concrete for punching is given by

$$V_{rd} = 0.25 f_{ctd} K_1 K_2 U d$$

Where U is perimeter of punching and its value is

For rectangular column type $U = 2(3d + L_c) + 2(3d + b_c)$

For circular column type $U = \pi(3d + c_{diam})$

Then check the value of V_{rd} with available punching shear value if it is greater or not. If $V_{rd}$is greater than that of V_P the depth does not require revision. The available punching shear value is computed as follows

$$V_P = (A_{total} - A_{punching}) * q_{avg}$$

Where $A_P$area of punching and its value is is

For rectangular column type $A_P = (3d + b_c)(3d + L_c)$

For circular column type $A_P = {}^{\pi}/_{4} * (3d + c_{diam})^2$

If the depth inserted by the user is failed for both punching and wide beam shear the system will notify the user to modify his input and by changing his input the user can try the design again.

17.5. Determination of Depth

If depth of the footing is not given, the program itself will find the depth which satisfies both punching shear and wide beam shear by taking an initial assumption. The initial assumption is made depending on the minimum requirement of EBCS for footing rested on soil which is 150mm above bottom reinforcement. (EBCS sec. 6.5.5.1)

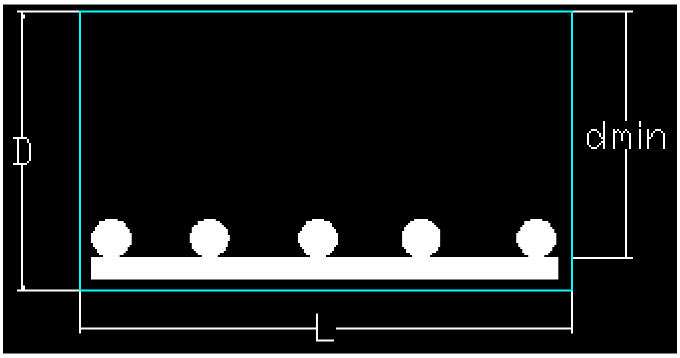


Figure 17-1

The gross depth would be then

$$D = 150 + \emptyset_L + cover$$

Depending on the assumed gross depth, effective depths in both B and L direction will be computed in a manner of

$$d_{effL} = D - cover - {}^{\emptyset_L}/_{2}$$

$$d_{effB} = d_{effL} - {}^{\emptyset_L}/_{2} - {}^{\emptyset_B}/_{2}$$

Where $\emptyset_L$ and $\emptyset_B$ are bar diameters which are extended along L and B direction respectively. The user has two ways for selecting those bar diameters. The first one is by inserting minimum and maximum diameter and the other one is inserting one diameter for both directions. If the user selects maximum and minimum diameter, the system will sets different bar combinations and checks which one is more economical within the given maximum and minimum spacing. For the first trial the diameters in B and L direction will be taken as the minimum bar diameter selected by the user. In order to find the most economical bar diameter we followed the procedure of calculating total weight of different bars combination and the combination which has less weight would be selected. The procedure goes like this by interchanging diameters in B and L direction and the final selected bar will appear on the design output window.

The user can also use the second option of inserting one bar diameter for both directions, d_{effL} and d_{effB} will be computed using that diameter. But for example, if the diameter makes the reinforcement arrangement congested, there will be a notification to the user to modify his preference.

The average effective depth would be then;

$$d_{effAvg} = \frac{d_{effL}+d_{effB}}{2}$$

The depth is going to be approved by checking if it satisfies the requirements of punching shear and wide beam shear. If the depth does not satisfy the requirement the initial assumption should be revised by adding its half on the previous checked depth. But here, one thing we have considered is that when adding D/2 in the previous failed depth to proceed to the next trial and incase if the incremented depth becomes safe for both wide beam and punching shear, it doesn't mean that depth is going to be selected. Because there is another checking of depth between D and D/2 if there is an appropriate depth between them. Then by reducing the factor to be added the system go to another trial till it is satisfied and economical. By successive iteration, finally we calculate the depth correct to 0.1mm accuracy.

17.6. Reinforcement Calculations

Here, the required area of steel is calculated in both directions independently. The area of steel would be then

$$A_s = \rho b d$$

While calculating the geometrical ratio of reinforcement, it needs to be checked with minimum reinforcement ratio as it is stated on EBCS. If the calculated reinforcement ratio in each direction of L and B is less than minimum requirement ($\rho_{min} = {}^{0.5}/_{f_{yk}}$) we take the minimum value and calculating area of steel goes on.

Therefore, geometrical ratio in L and B direction are calculated as:

$$\rho_{LL} = \frac{1 - \sqrt{1 - \frac{M_{LL}}{f_{cd} * L * d_{effL}^2}}}{{}^{f_{yd}}/_{f_{cd}}}$$

$$\rho_{BB} = \frac{1 - \sqrt{1 - \frac{M_{BB}}{f_{cd} * B * d_{effB}^2}}}{{}^{f_{yd}}/_{f_{cd}}}$$

M_{LL} and M_{BB} are calculated at the critical section on the footing. As the software designs only footings which support concrete column, the critical section is located at the face of the column. (EBCS section 6.5.1) they are determined as

$M_{LL} = {}^{1}/_{2}\, q_u * B * ({}^{L}/_{2} - {}^{l_c}/_{2})^2$ In L direction and

$M_{BB} = {}^{1}/_{2}\, q_u * L * ({}^{B}/_{2} - {}^{b_c}/_{2})^2$ In B direction

17.7. Detailing

In this step the process is mostly dependent on the data inserted by the user. If the user selects one spacing for both direction the band and outside of the band spacing can be a factor of the selected spacing.

$$s_b = \frac{2}{\beta * 1} * S \qquad s_{ob} = \frac{\beta + 1}{2} * S$$

But if the user selects one diameter for both directions the spacing will be calculated as below and the band and outside band spacing will be a factor of the calculated spacing

$$S = \frac{a_s}{A_s} * 1000$$

After providing the bar using the calculated spacing we should check the design outputs. The software designs the footing in the most economical way and as it is a computer program the design can be made till the previous calculated values of depth and reinforcement ratios are equal with the new calculated values.

17.7.1. Checking for Anchorage and Hook Length

This is the last step of the designing procedure. This is order to check the bar type if it is straight or hooked by calculating anchorage of reinforcement. In calculating the anchorage of reinforcement, the bottom reinforcement measurement depends on the projection of the footing from the critical section for moment. In our case, as the column is concrete column and its critical section is at the face of the column, we need to check whether it exceeds the effective depth at that section or not in each direction. (EBCS sec 6.5.2.3 & 6.5.2.4)

If the projection at critical section exceeds the effective depth at that section:

$$l_{avail} = \frac{L \text{ or } B}{2} - \frac{l_C or b_C}{2} - cover$$

If it doesn't exceeds

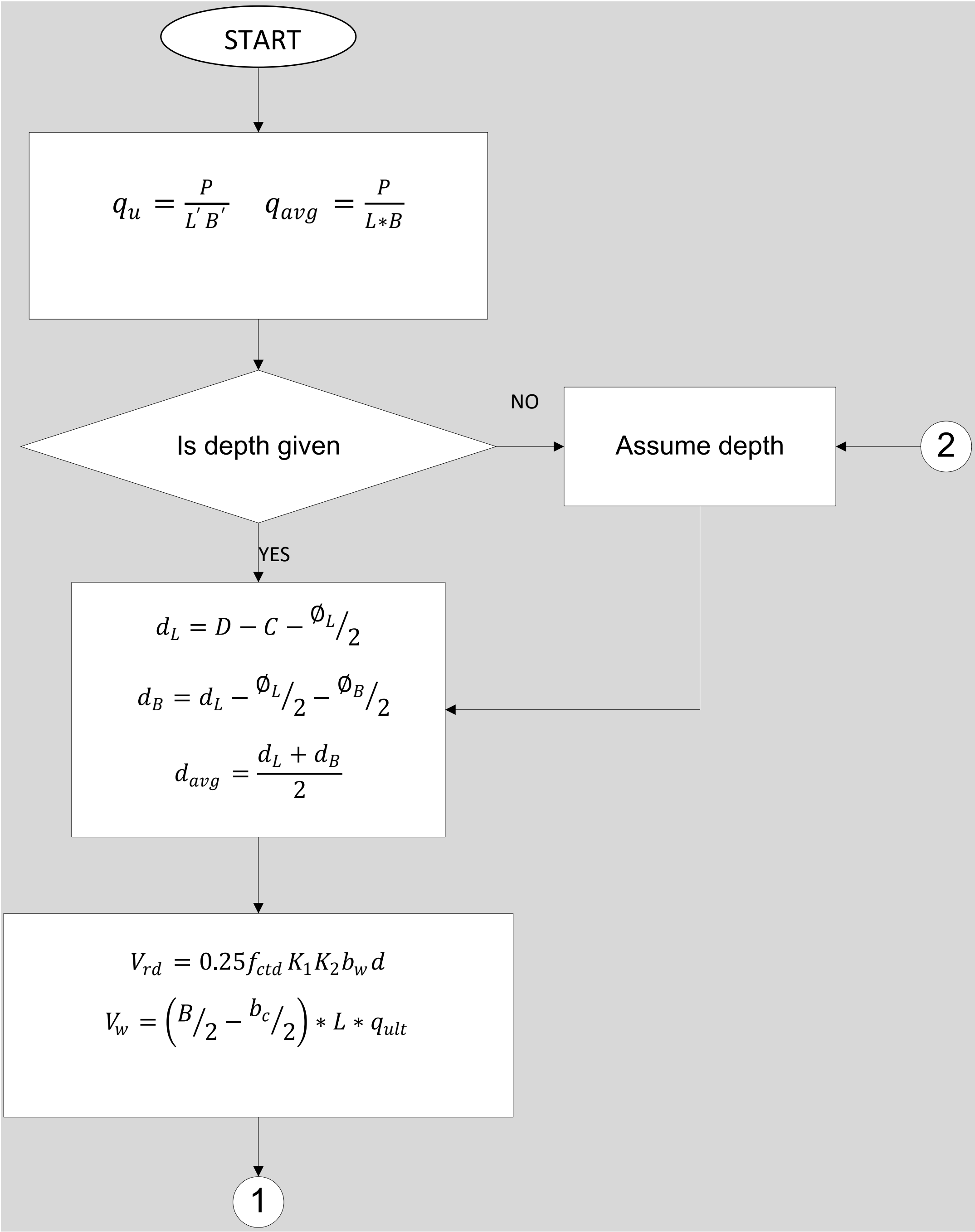
$$l_{avail} = \frac{L \text{ or } B}{2} - \frac{l_C or b_C}{2} - d - cover$$

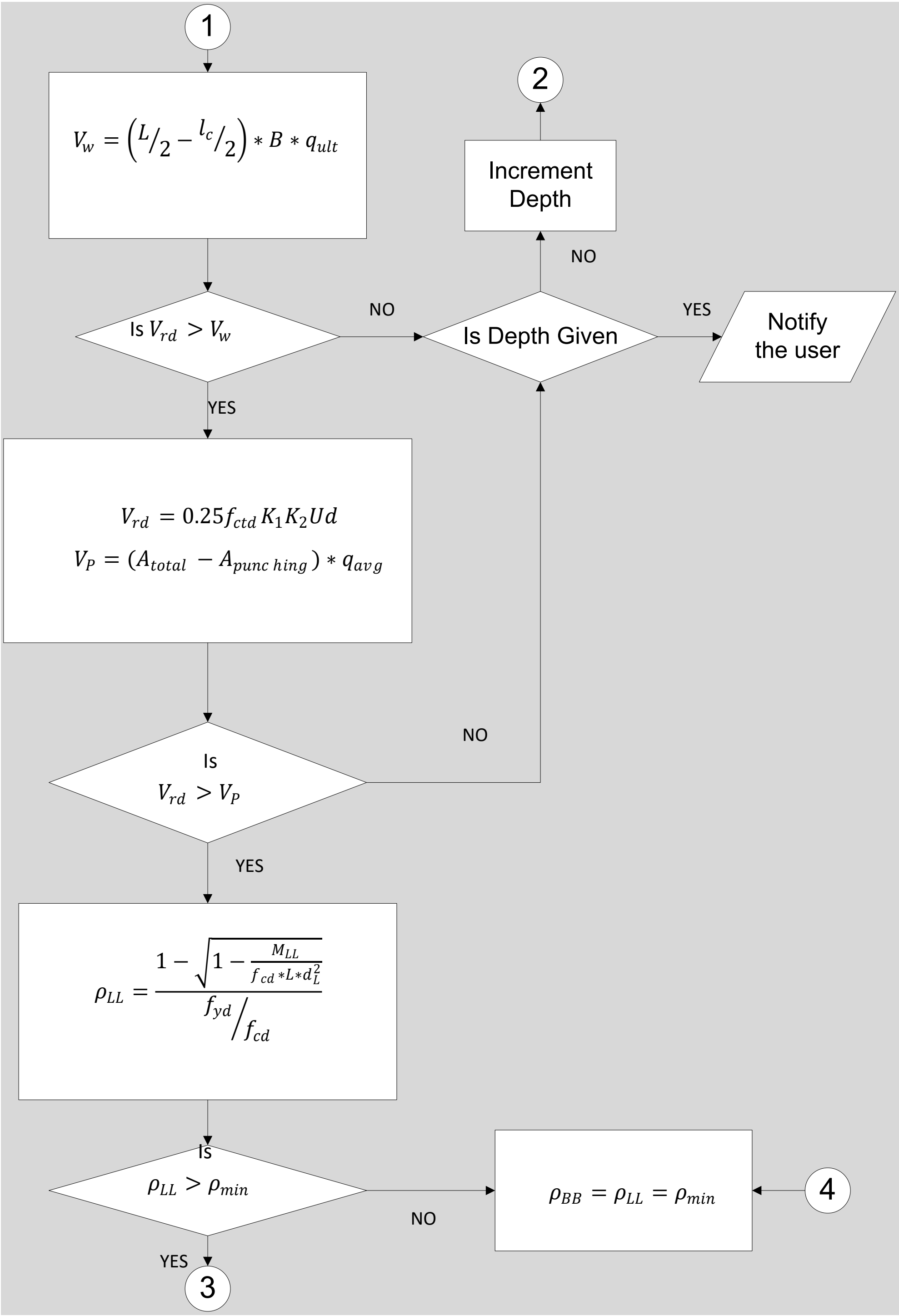
After calculating the available anchorage length it should be checked with net anchorage length and if it is greater we use straight bar otherwise we use hooked bar type. (EBCS sec 7.1.6)

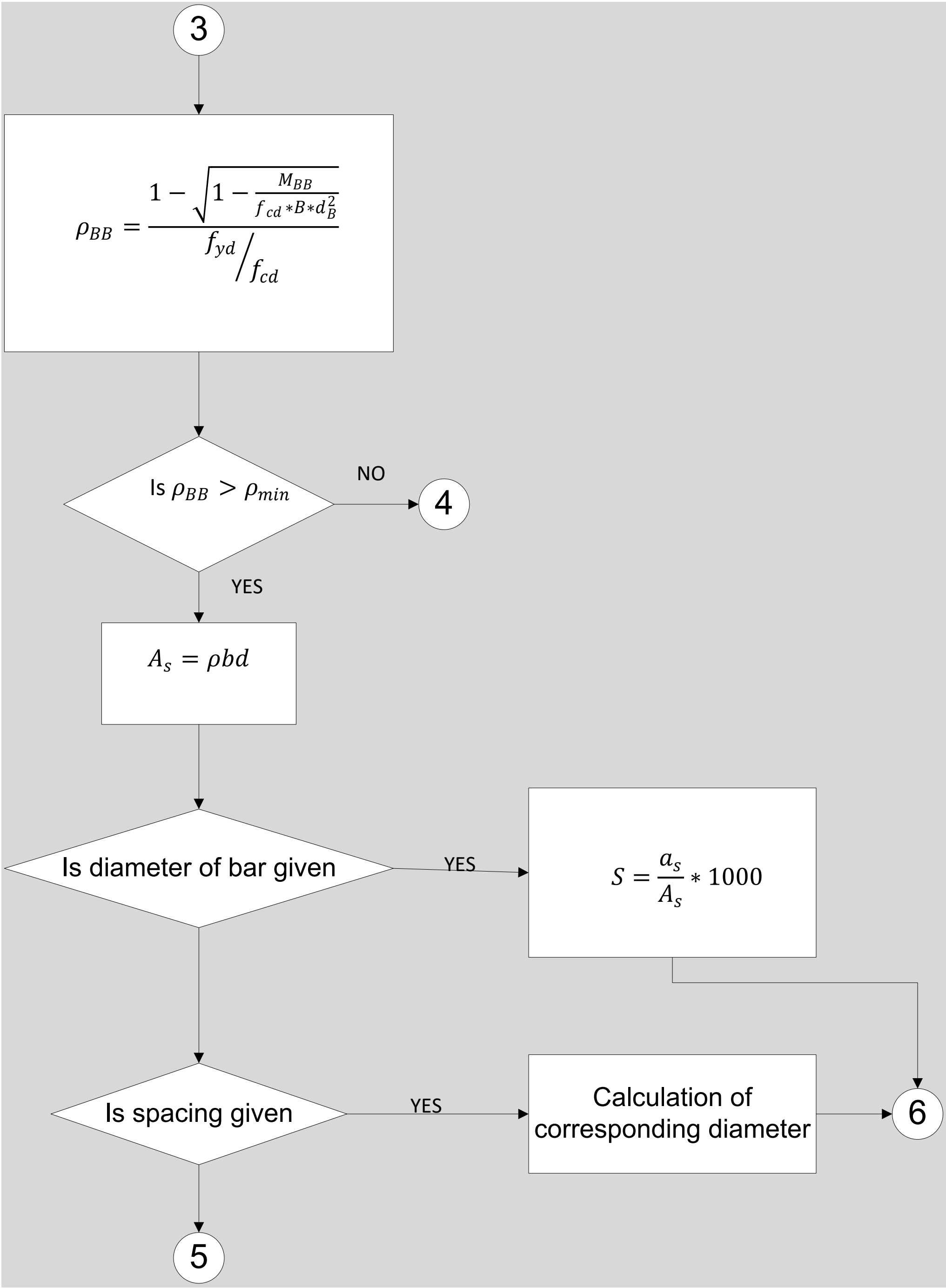
$$l_{bNet} = al_b \frac{A_{scal}}{A_{sprov}} \geq l_{bmin} \qquad l_b = \frac{\emptyset}{4} * \frac{f_{yd}}{f_{bd}} \qquad f_{bd} = 2 * f_{ctd} \ for\ deformed\ bar$$

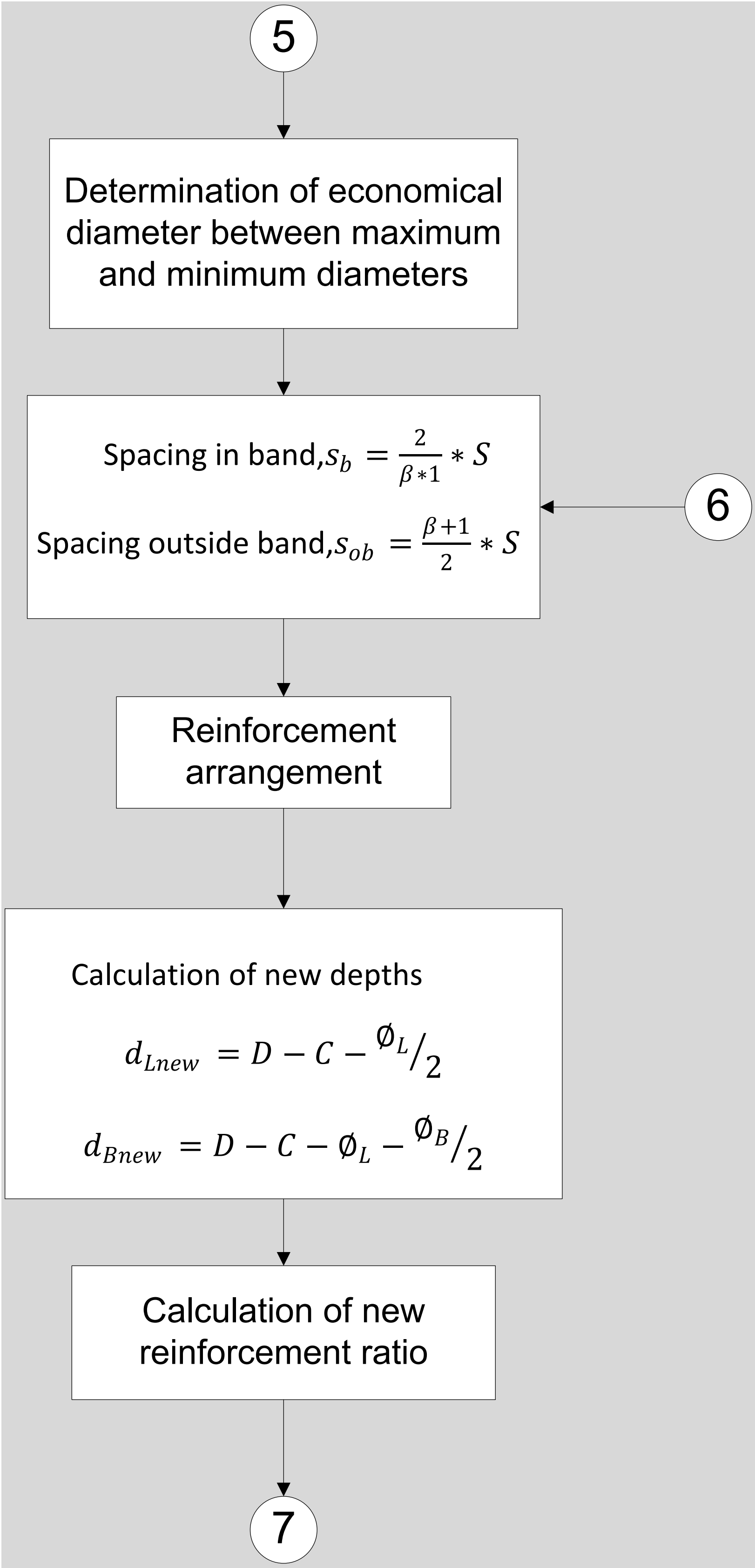
17.8. Flow Chart

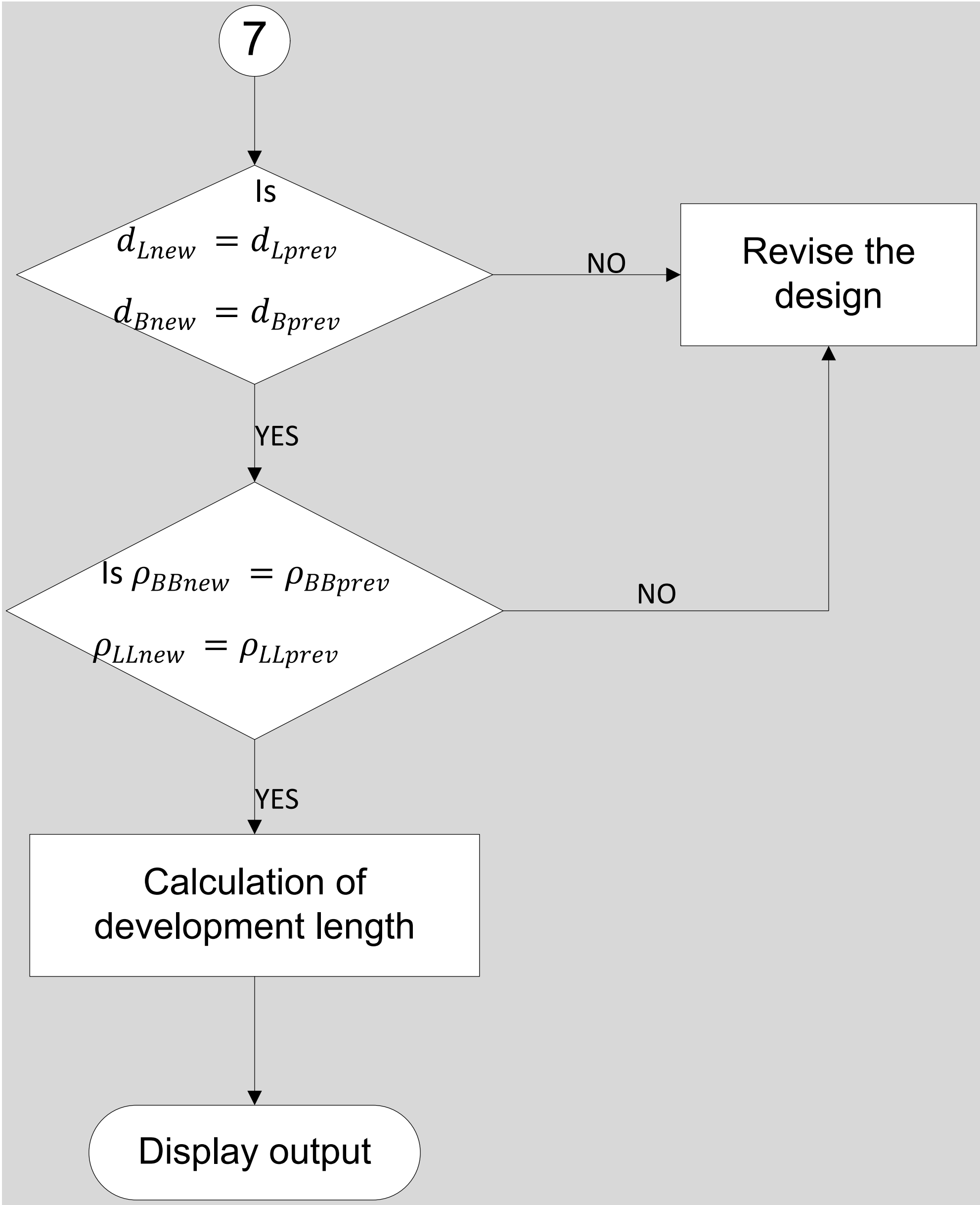
Flow charts are drawn independently for general and detailed procedure because in the general procedure flowchart the reader can easily understand the general idea about the procedure we followed. In the detailed procedure flow chart formulas of different parameters are included and here also the reader can have the idea of how the calculations are done in the developed program.


START
$q_u = \frac{P}{L' B'}$ $q_{avg} = \frac{P}{L*B}$
Is depth given
NO
Assume depth
2
YES
$d_L = D - C - {}^{\emptyset_L}/_2$
$d_B = d_L - {}^{\emptyset_L}/_2 - {}^{\emptyset_B}/_2$
$d_{avg} = \frac{d_L + d_B}{2}$
$V_{rd} = 0.25 f_{ctd} K_1 K_2 b_w d$
$V_w = \left({}^{B}/_2 - {}^{b_c}/_2\right) * L * q_{ult}$
1


1
$V_w = \left(L/2 - l_c/2\right) * B * q_{ult}$
2
Increment Depth
NO
Is $V_{rd} > V_w$
NO
Is Depth Given
YES
Notify the user
YES
$V_{rd} = 0.25 f_{ctd} K_1 K_2 U d$
$V_P = (A_{total} - A_{punc\,hing}) * q_{avg}$
NO
Is $V_{rd} > V_P$
YES
$\rho_{LL} = \frac{1 - \sqrt{1 - \frac{M_{LL}}{f_{cd} * L * d_L^2}}}{f_{yd}/f_{cd}}$
Is $\rho_{LL} > \rho_{min}$
NO
$\rho_{BB} = \rho_{LL} = \rho_{min}$
4
YES
3


3
$\rho_{BB} = \frac{1-\sqrt{1-\frac{M_{BB}}{f_{cd}*B*d_B^2}}}{f_{yd}/f_{cd}}$
Is $\rho_{BB} > \rho_{min}$
NO
4
YES
$A_s = \rho bd$
Is diameter of bar given
YES
$S = \frac{a_s}{A_s} * 1000$
Is spacing given
YES
Calculation of corresponding diameter
6
5


5
Determination of economical diameter between maximum and minimum diameters
Spacing in band, $s_b = \frac{2}{\beta * 1} * S$
Spacing outside band, $s_{ob} = \frac{\beta + 1}{2} * S$
6
Reinforcement arrangement
Calculation of new depths
$d_{Lnew} = D - C - {}^{\emptyset_L}/_2$
$d_{Bnew} = D - C - \emptyset_L - {}^{\emptyset_B}/_2$
Calculation of new reinforcement ratio
7


7
Is $d_{Lnew} = d_{Lprev}$ $d_{Bnew} = d_{Bprev}$
NO
Revise the design
YES
Is $\rho_{BBnew} = \rho_{BBprev}$ $\rho_{LLnew} = \rho_{LLprev}$
NO
YES
Calculation of development length
Display output

18. Development – Footing

Outlines

- ❑ Use Cases
- ❑ Object Oriented Modeling

This chapter shortly describes the whole software development procedure used to produce functional footing design component. The development process starts by preparing Use Cases, which allow scenario based modeling for the software in its graphical user interface (GUI). Then, object oriented modeling is done. Object oriented modeling involves manipulation of operation, expected relation between objects and other object oriented programing issues. In this topic we only describe the most basic issues in software development procedure. Since, the expected readers of this document are Civil Engineering Professionals; more emphasis is given to technical part related to Civil Engineering. But, as a software development project every development procedure and documentation should be described. Therefore, the need of this chapter would be more critical for the sack of completeness only. Software Engineering extremely wide, describing every steps and procedures of development take long. Considering this we limit ourselves only in very important topics.

18.1. Use cases

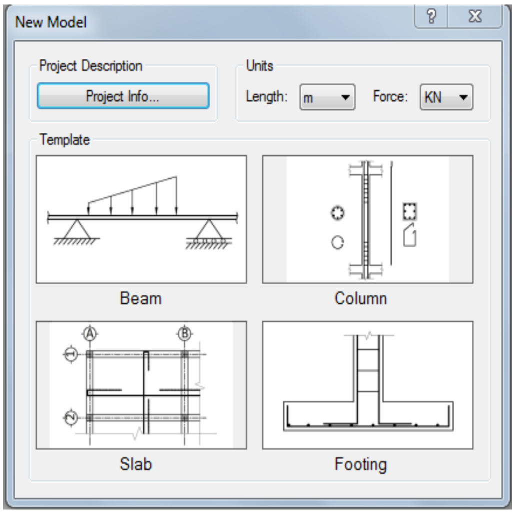
Actor: user

Goal in context: to design an isolated footing

Preconditions: the user must insert all the necessary data to start the design

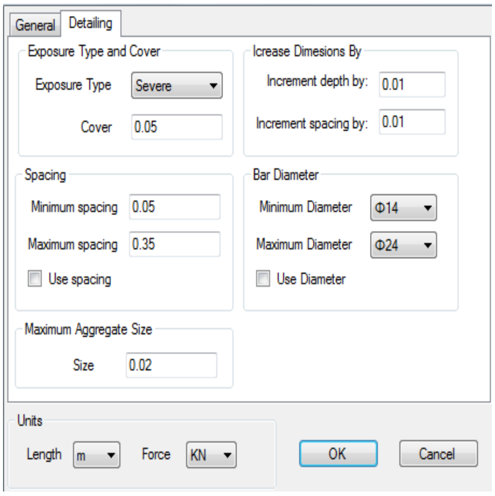
Scenario:

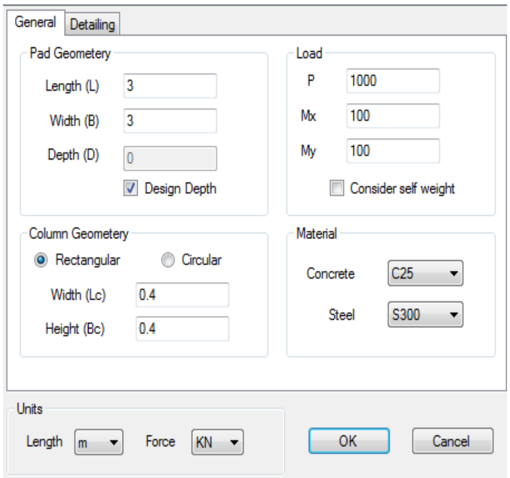
1. From file menu the user selects new model
2. The new model dialog box will appear on the ESADS window and then from those components listed on than dialog box, the user can select footing by clicking on the diagram of footing.



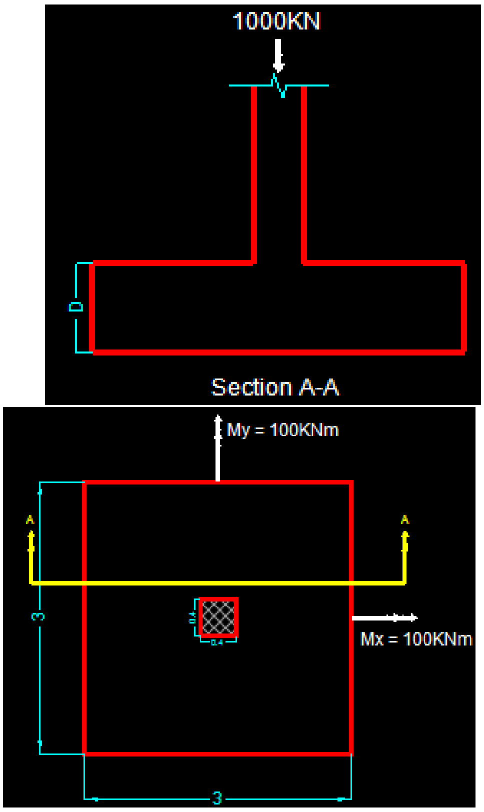
3. After the user clicks on footing, in the general dialog box appeared the user can insert the required data to run the design and the user can also adjust his bar preference and information related to dimension increment factors by clicking on detailing dialog box

4. The user clicks ok button to proceed or if he wants to exit the user clicks on Cancel button.

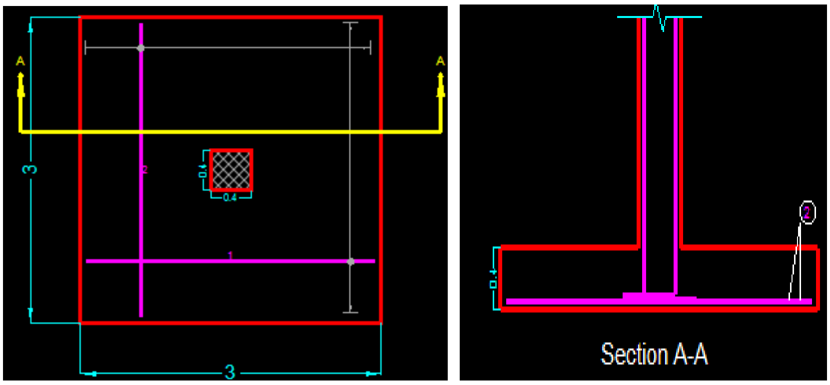


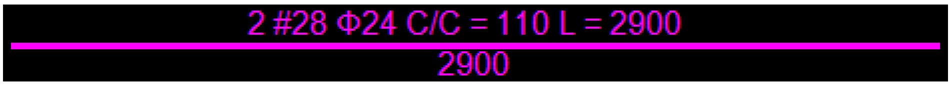


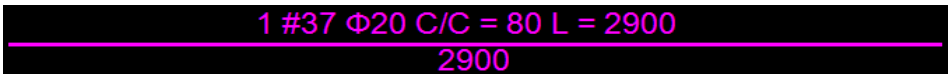
5. If the user clicks on OK button the system will shows preview of the footing. So, the user can check the diagram of the footing if it is according to the data he inserted on the previous dialog box.



6. From the Run menu the user can run the design by selecting “Run design” option or just by clicking on the short cut “F6”.If the data inserted by the user are sufficient enough for the design the user can see the design output which displays in a speed of someone takes to blink his eyes.







Exceptions: If the design is failed because of insufficient depth or user’s bar preference, the design process will be stopped and tells the user to modify his inputs.

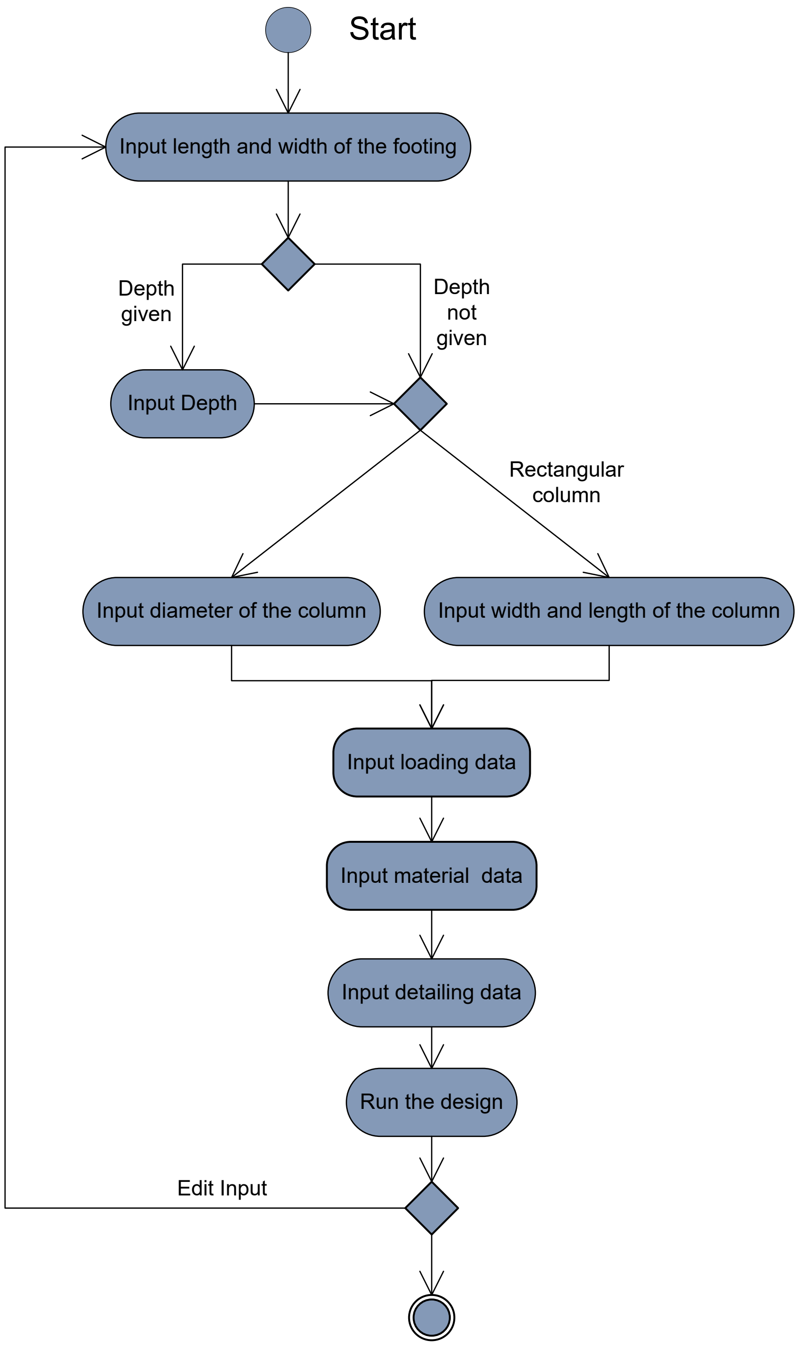


Figure 18-1 Activity diagram

19. Recommendation

Better to do it this way!!

Outline:

- ❑ To Students
- ❑ To Developers
- ❑ To Educational Institutions

This is a short chapter to give a way for future developers, students and higher educational institutions. The project team members are giving these recommendations from their experience and thought.

19.1. To Students

Most senior projects in civil engineering at the level bachelor degree are design projects, specifically on building and highway. Designs are good enough to think up to the extent to where they learnt in class. However, in senior projects, real critical thinking beyond what they have learnt. We strongly believe that students should defend their new idea, method or theory at the end of their school year, not an implementation of an already existing procedures.

Therefore, we strongly recommend future students to think of something that really solves a problem of the society. An excellent work would be to solve a real problem not that has already been solved.

As we mentioned in section 1.9, we faced many challenges. No matter how we suffered, we managed to achieve a considerable result. We had to read a lot, spend nights, trying to see how a block of code works. We reached here just because we promised not to back-down until we reach our goal. We had a promise not halt the work even if no one is going to like or appreciate it. In short "We dreamed in desert and ended up in a green city."

Therefore, for those of you who have new idea, even if it seems silly, try to improve it and prove its feasibility to yourself. Base your proof on reality and scientific ways so that you will not spend your time in impossible work. Then try to show the people around you to show what you saw.

Appreciation has a great positive impact on your motivation for your work. On the contrary, pessimistic comments may create huge destruction for your courage in continuing the work. When hearing to comments of others, it is advisable to hear the fact, not the way they understand it. This helps in avoiding the comment of people who foreword their comment before understanding the issue.

19.2. To Developers

Due to limitation of ability and resource, we didn't conduct a formal requirement analysis and research. This would have been very nice way to decide what exactly the current burning issue of the designers.

Anyone who is planning on developing similar software is recommended to conduct a formal research and requirement analysis so that the most important problems will be addressed first. In the requirements analysis, designers may even tell new ways of the working algorithms that may be used in the design of the software.

Furthermore, we didn't have a highly organized documentation. We always rushed to the coding before exhaustively modeling the analysis and architectural design of the software. These models are very helpful to work in very organized way.

Documentation is a very important portion of software development. It enables the developer to familiarize back when returning to the work after sometime.

19.3. To Educational Institutions

One of the challenges we faced was lack of understanding of the higher officials of the campus. We needed to spend a summer working on the project by consulting experts, which they were not willing to provide. They may have their own internal problem not to provide what we requested, but they didn't do the least thing – willingness to checkout our proposal and see what we are trying to do. That was really disgusting for us, sacrificing a full summer not doing so much as we planned.

Educational institutions are the sources of every intellectuality. In order to keep this fact they have to work continuously in by appreciating the ideas created in them. New idea is like newly emerging seedling, once broken it takes a lot to heal it back.

Therefore, we strongly recommend higher educational institutions to improve their attitude towards new scientific ideas coming from their students.