Dedicated To

AYELE TESFAYE ADMASU

Our Physics teacher back in high school

And who encouraged us to start reading on Software Engineering

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

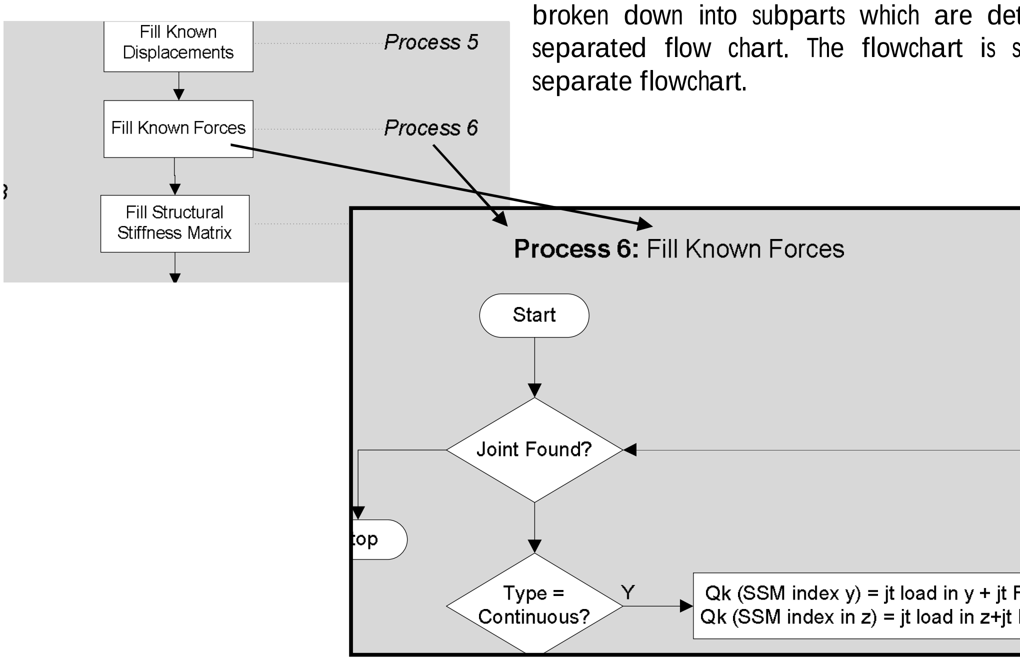
The document has a total of four main parts: ESADS in General, Continuous Beam, Column and Footing. The first part is concerned with stating the general issues which are not specific to any other component. Those things given in this part apply to all other parts as well.

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.

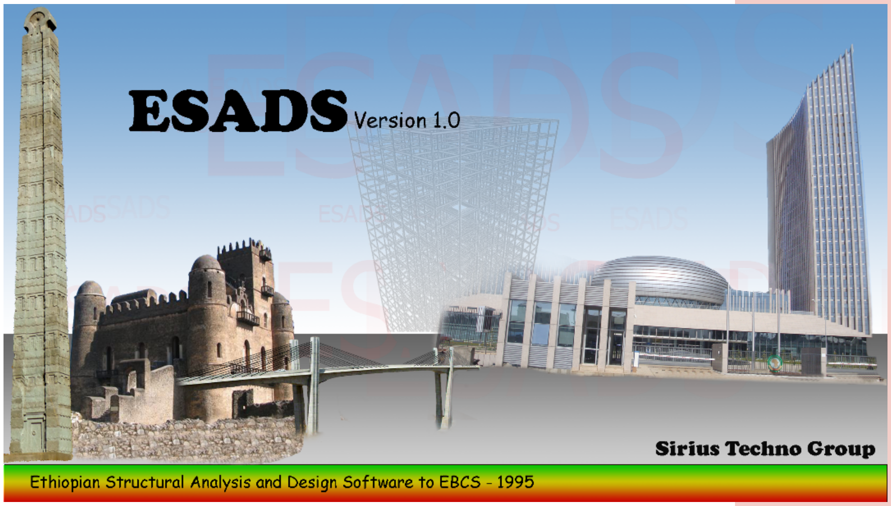
Visuals

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.



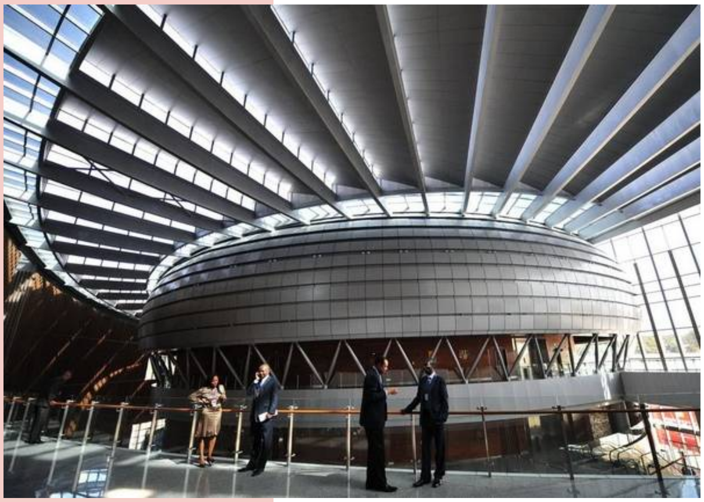
When it is important to use activity diagram, it is presented next to the corresponding use case. Most of the time activity diagrams are presented for complicated processes.

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).

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.

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.

- 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.

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.

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:

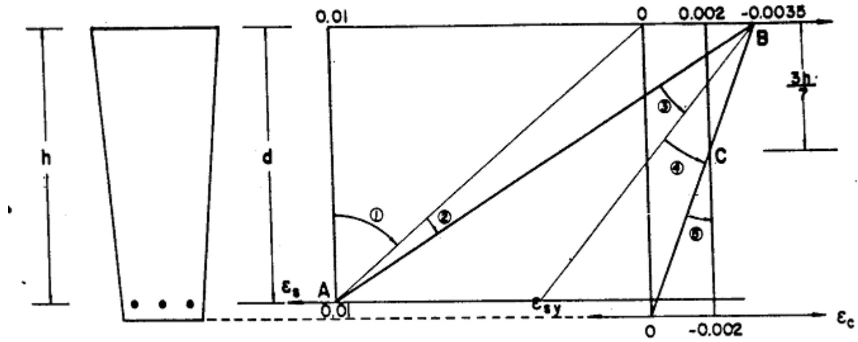


Figure 4.1 Strain Diagram in the Ultimate Limit State

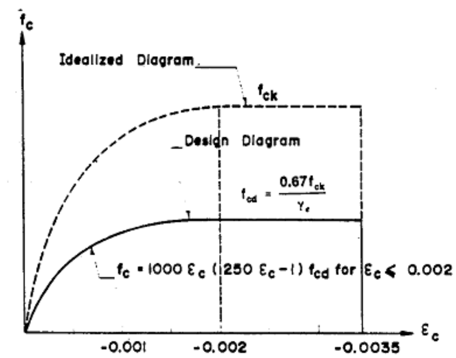


Figure 4.2 Parabolic-Rectanglar Stress-Strain Diagram for Concrete in Compression

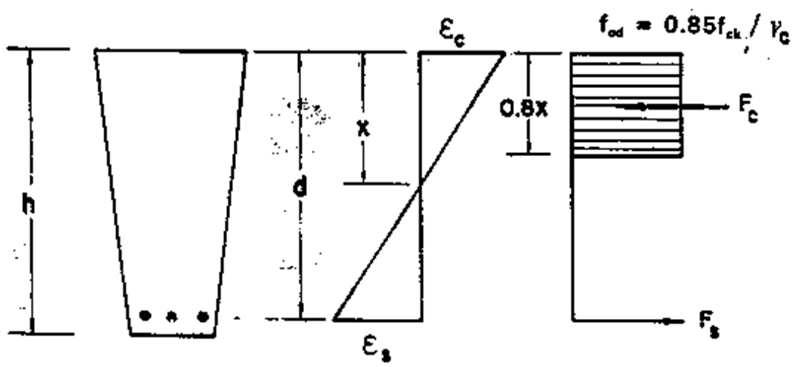


Figure 4.3 Rectangular Stress Diagram

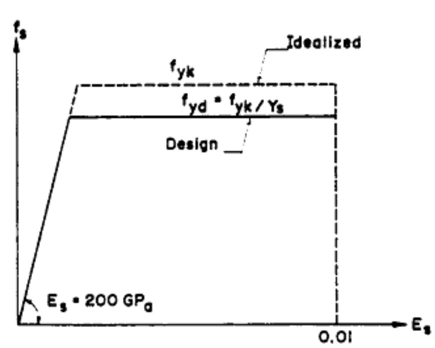


Figure 4.4 Stress-Strain Diagram for Reinforcing Steel

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

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

$$or \ \geq 200mm$$

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:

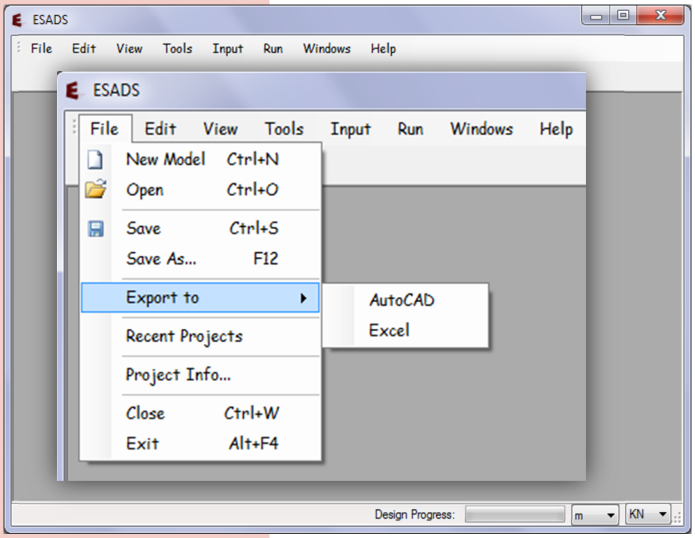
- Waterfall: a linear framework
- Prototyping: an iterative framework
- Incremental: a combined linear – iterative framework
- Rapid application development (RAD): an iterative framework
- Extreme programming

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.

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.

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.

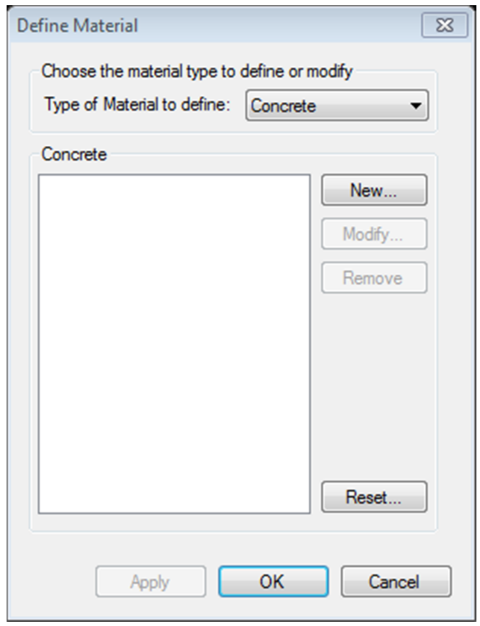


Fig 4-2: Define Material Dialog

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.

- 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.

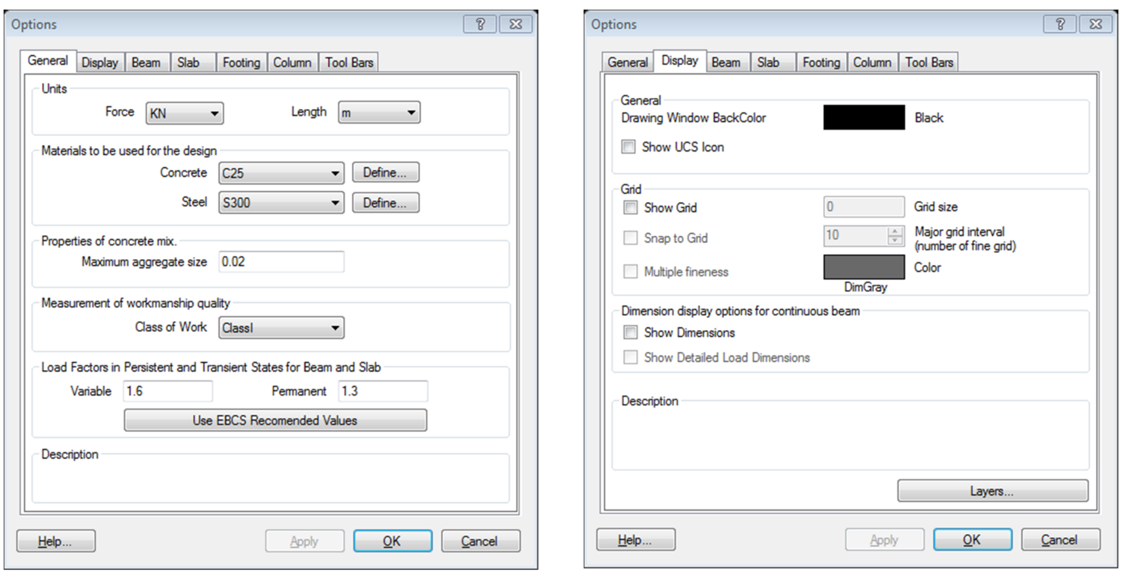


Fig 4-4: Options dialog with the General and Displays activated

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

5. Introduction to Beam

Outline:

- ❑ Overview
- ❑ Features
  - o Modelling and Input
  - o Member Input and Modification
  - o Joint Input and Modification
  - o Load Assignment and Modification
  - o Running the Analysis
  - o 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.

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.

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

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.

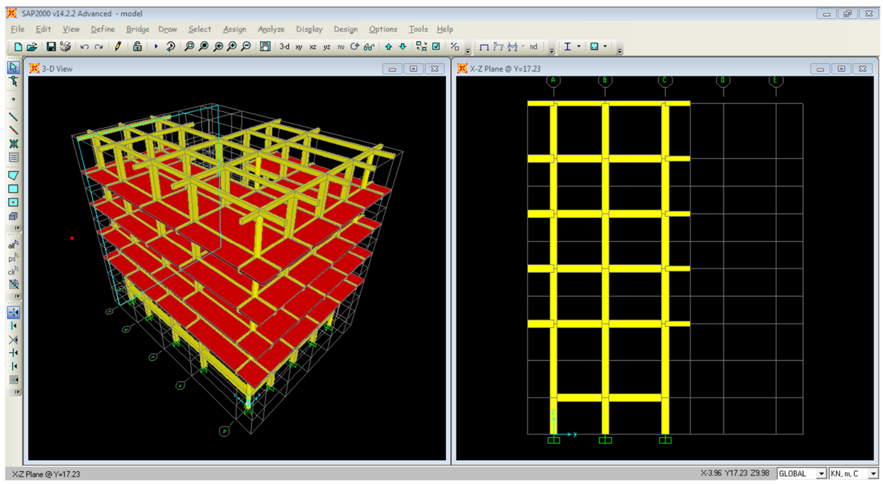
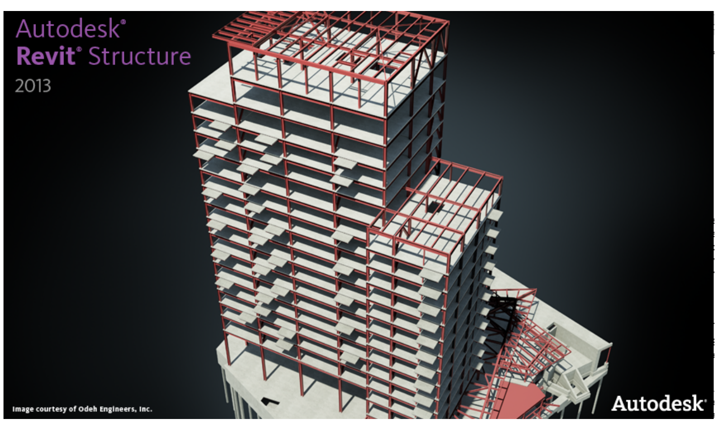


Figure 6-1: Model of a G+4 building done using SAP2000 v14.0

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.

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.

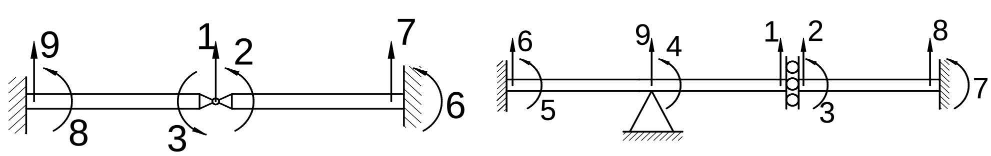


Fig. 7-1 Sample Degree of freedom nomenclature

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.

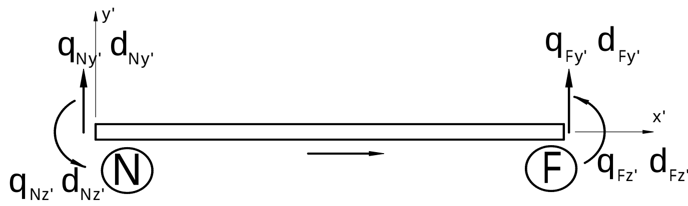


Fig 7-2 Positive Sign Convention

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.

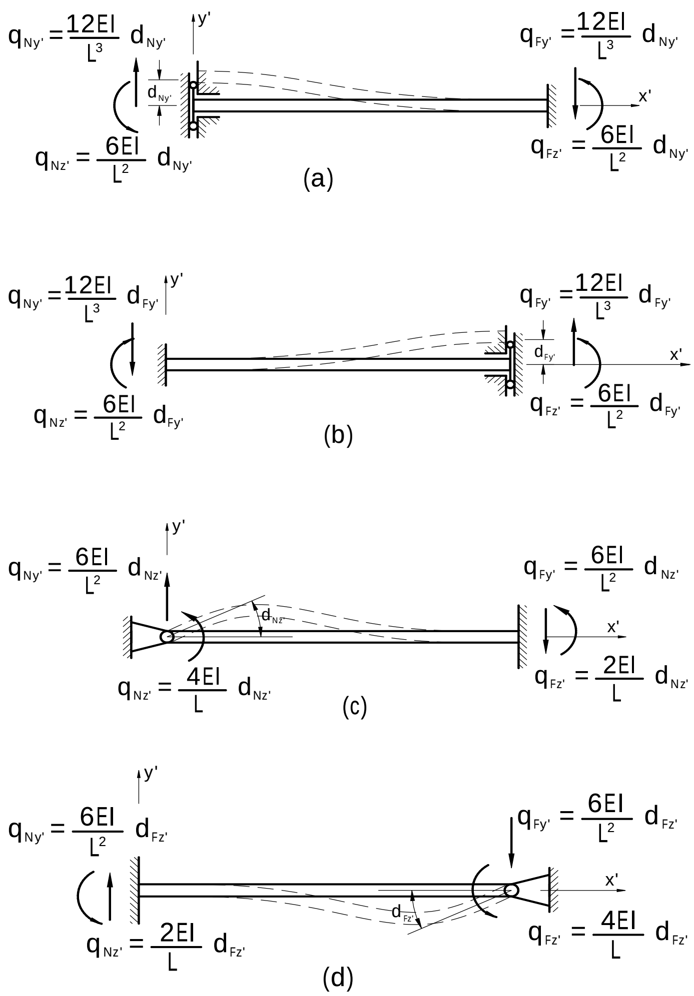
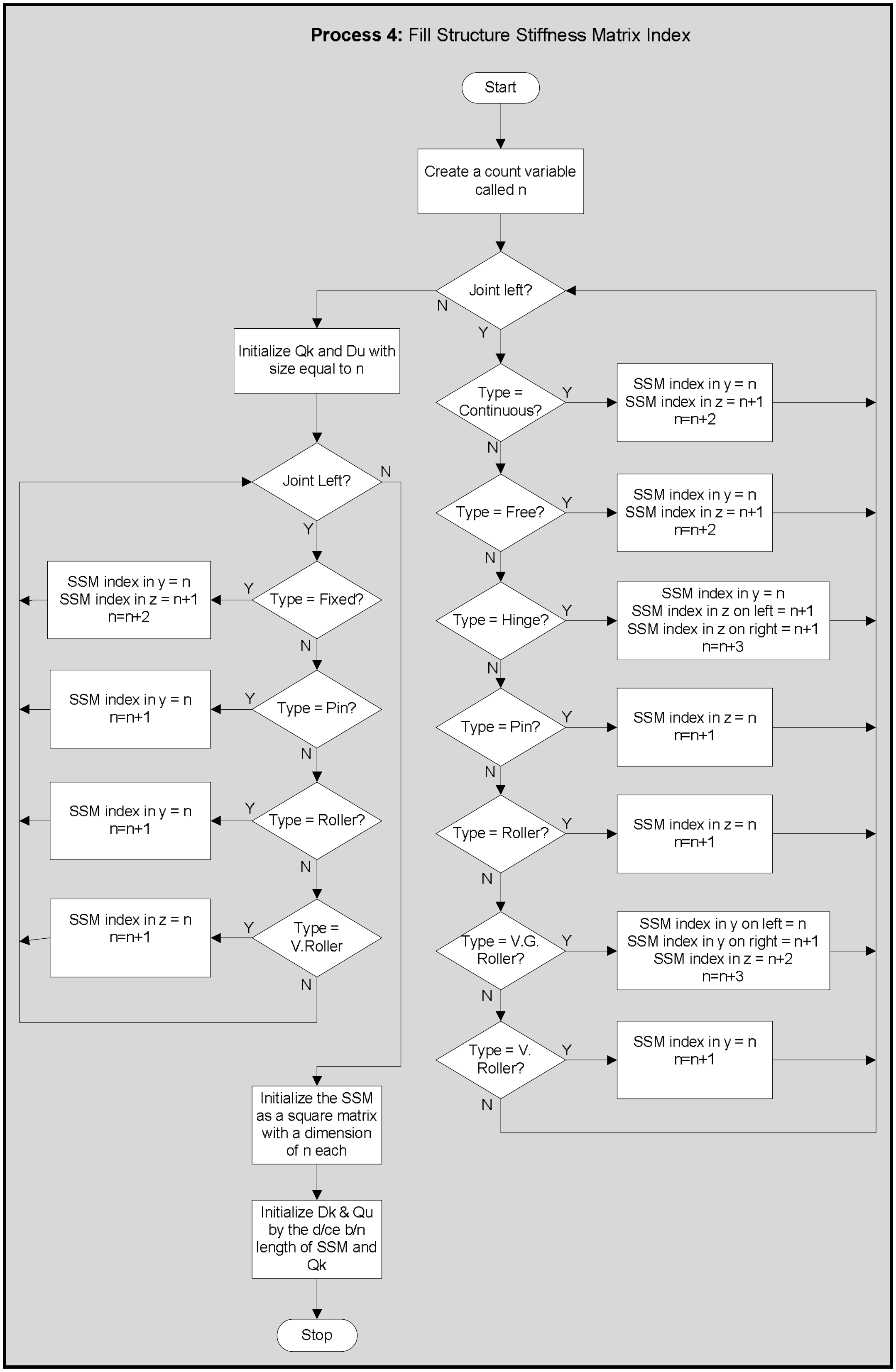


Fig 7-3: Displacements in y and z axis due to load

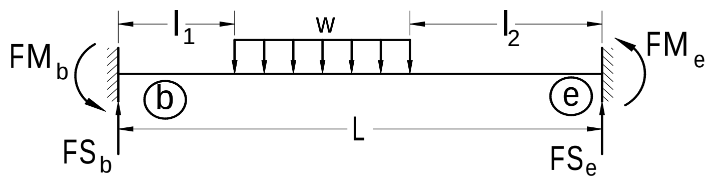
By superposition, if the above results in the above figures are added, the resulting four load-displacement relations for the member can be expressed in matrix from as:

Flow Chart 3: Filling Structure Stiffness Matrix Index



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

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



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

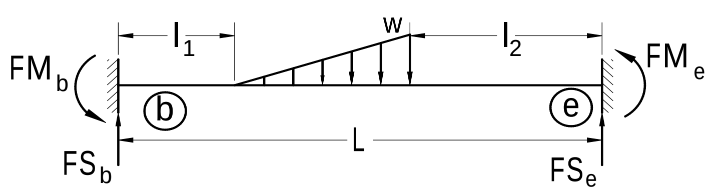
$$FM_b = \frac{wL^2}{12}\left[1 - \frac{l_1^2}{L^4}(6L^2 - 8l_1L + 3l_1^2) - \frac{l_2^3}{L^4}(4L - 3l_2)\right]$$

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

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

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.

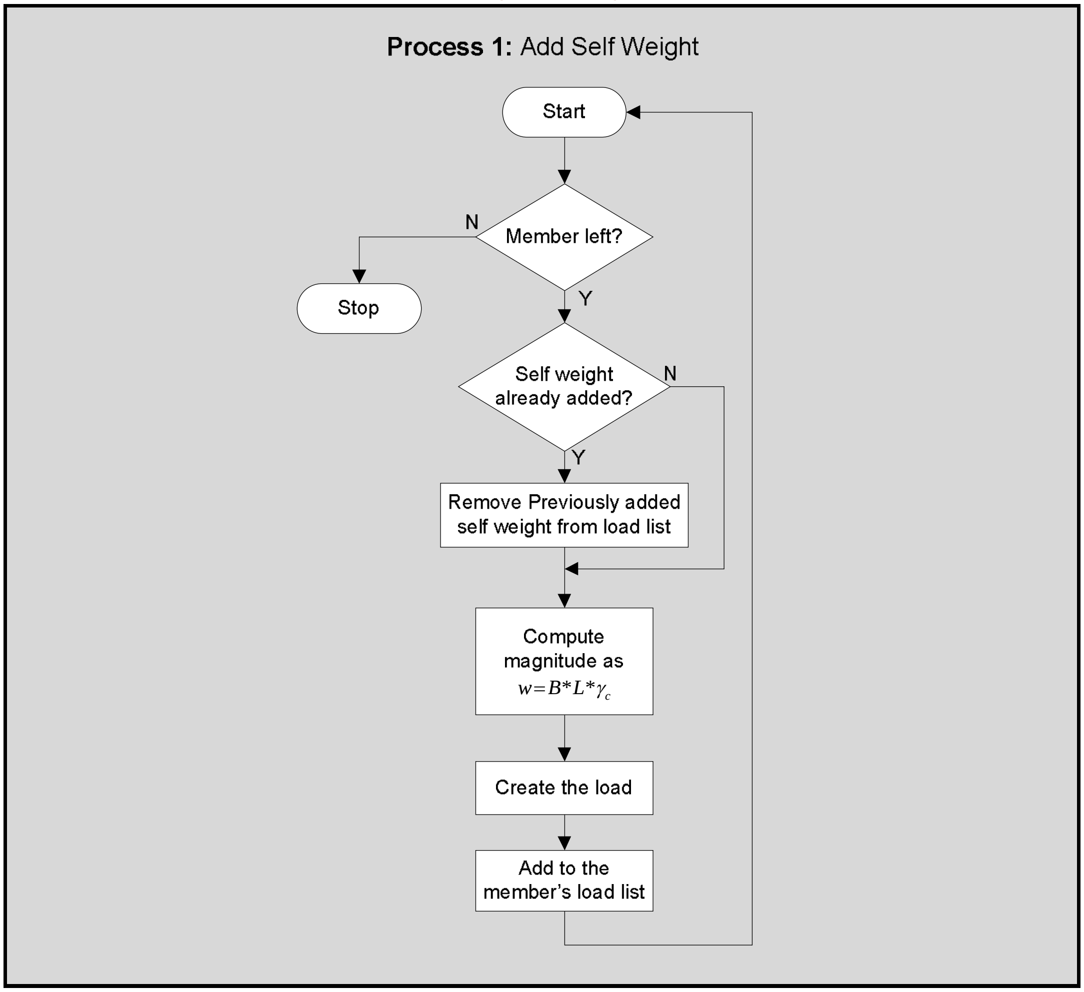


$$FS_b = \frac{w(L-l_1)^3}{60L^2}\left\{(3L + 2l_1)\left[1 + \frac{l_2}{L-l_1} + \frac{l_2^2}{(L-l_1)^2}\right] - \frac{l_2^3}{(L-l_1)^2}\left[2 + \frac{15L - 8l_2}{L-l_1}\right]\right\}$$

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

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

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

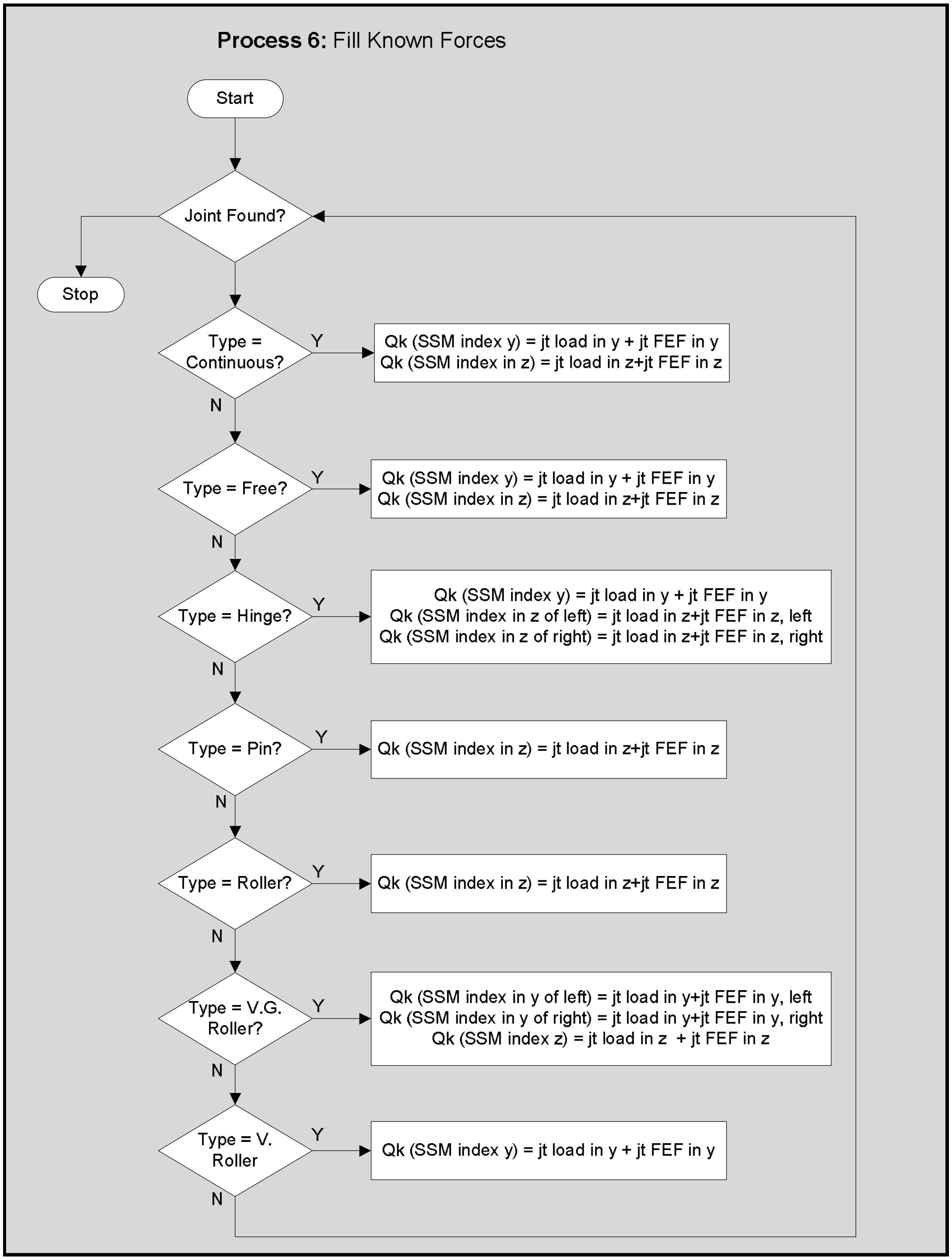
$$\begin{bmatrix} Q_k \\ Q_U \end{bmatrix} = \begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix} \begin{bmatrix} D_U \\ D_K \end{bmatrix} \qquad 7\text{-}4$$

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

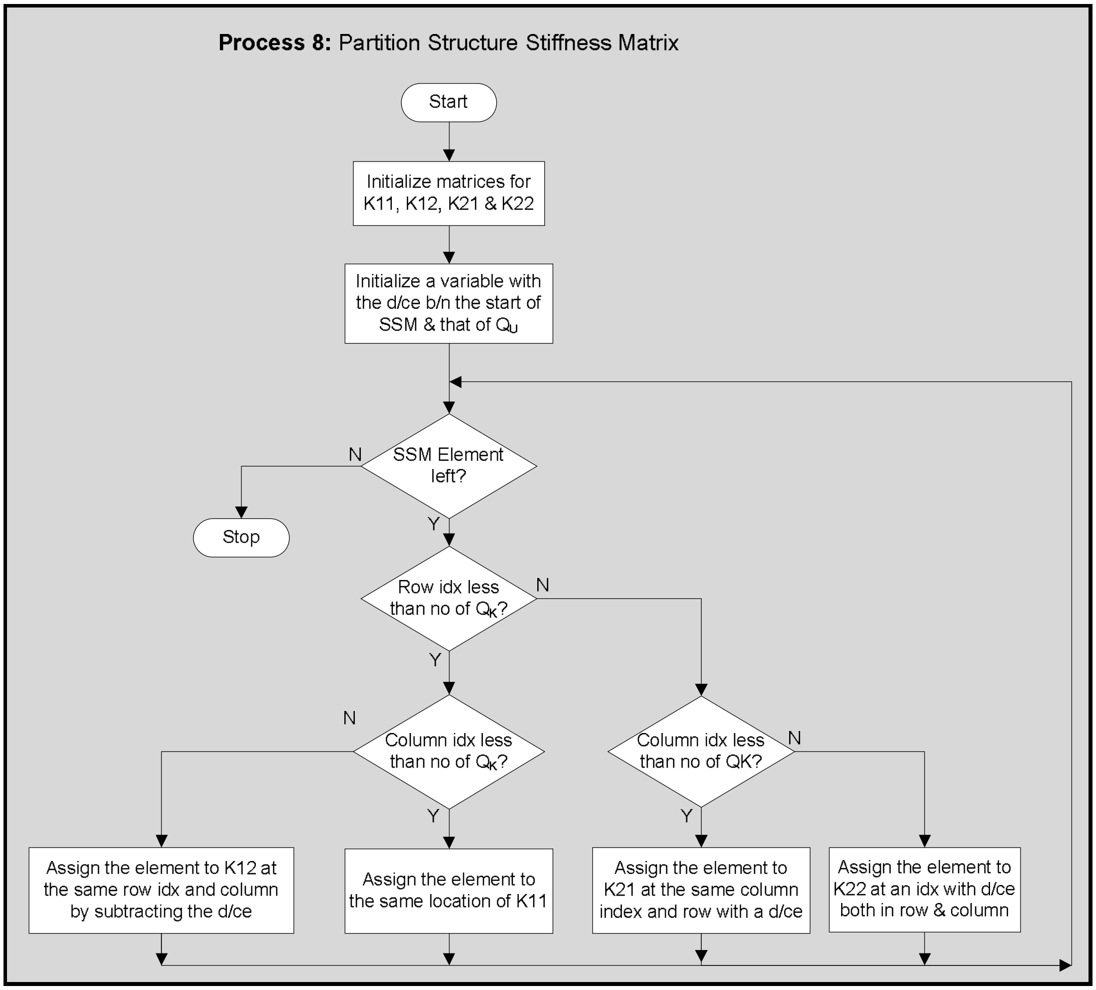
The above relation may be expanded as:

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

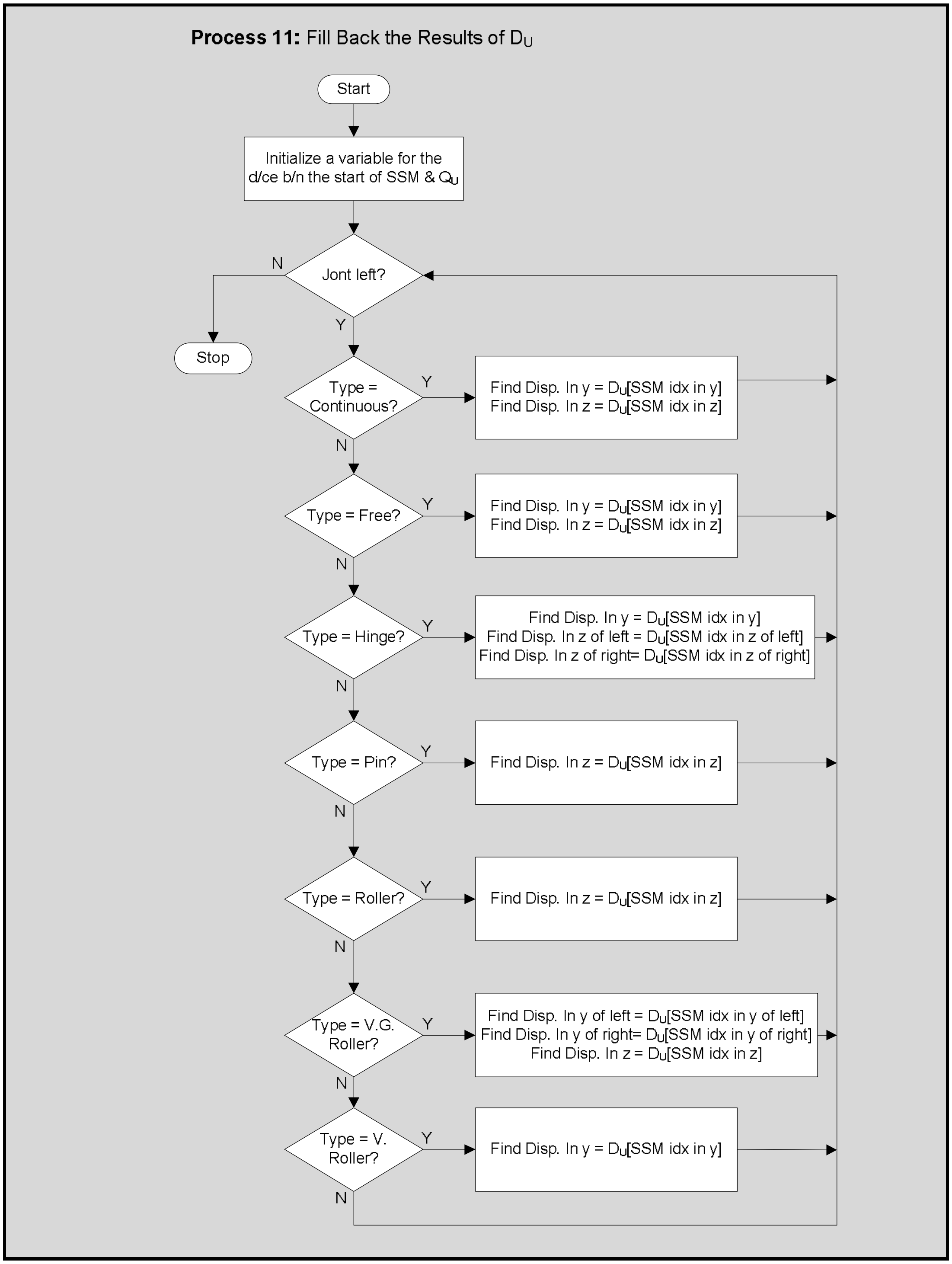
Flow Chart 7: Filling Known Forces



Flow Chart 9: Partitioning Structure Stiffness Matrix



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:

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.

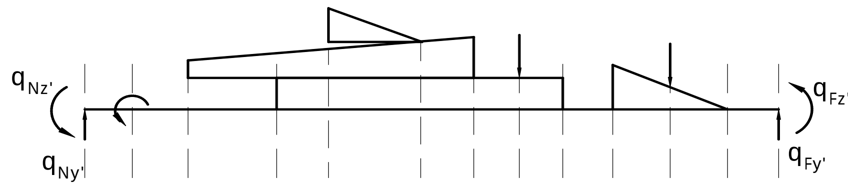


Fig 7-10: Intervals of loading which can be represented by a single polynomial expression.

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

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

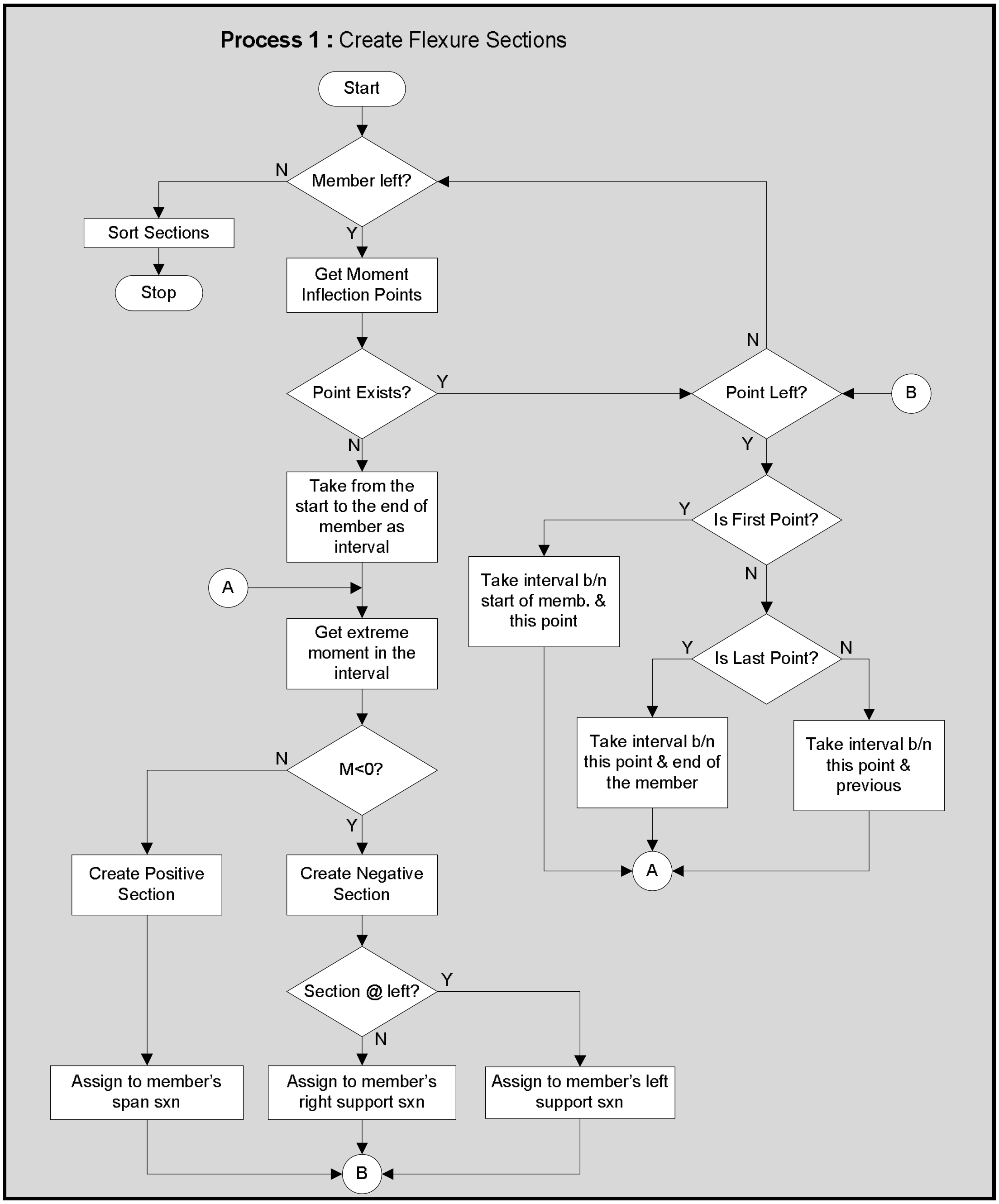
In the previous chapter, we have discussed the working mechanism of ESADS to analyze a continuous beam. Now we are going to discuss how ESADS uses the bending moment and shear force diagrams to design the beam.

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 15: Creation of Flexure Sections



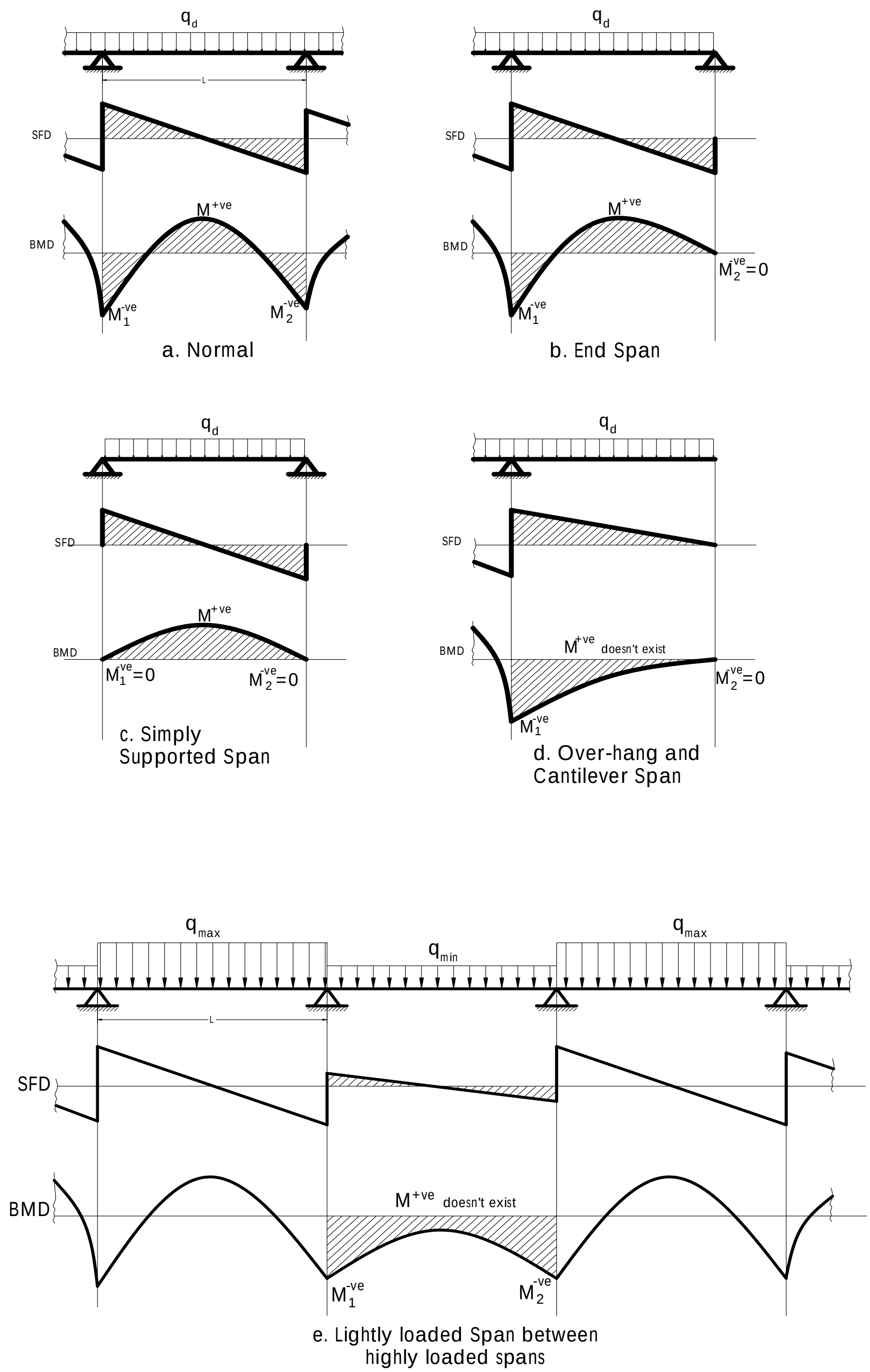
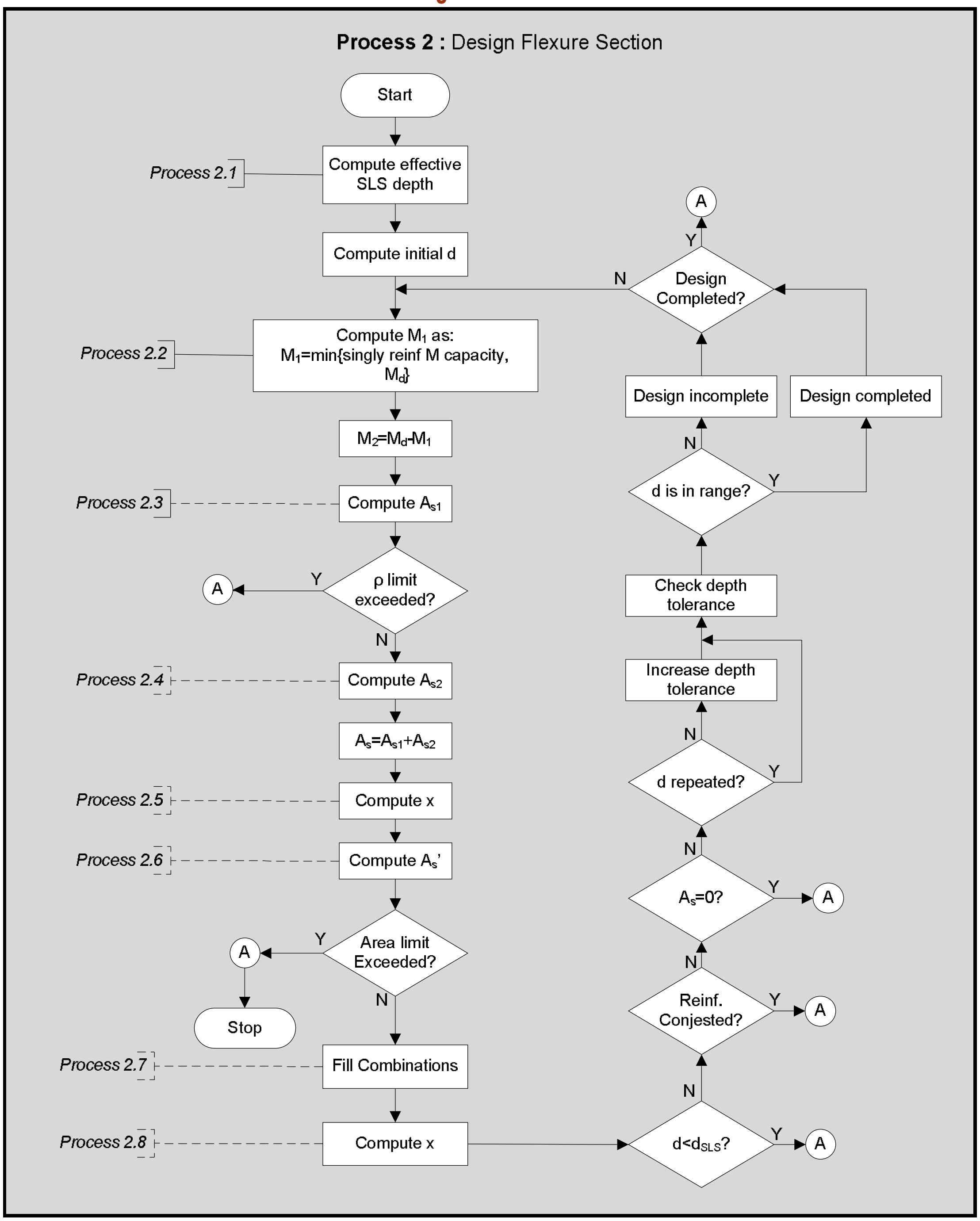


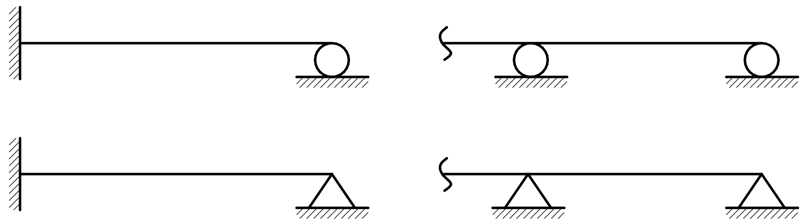
Fig 8-1: Moment diagram possibilities of common building beams

8.2. Design of Beam for Flexure

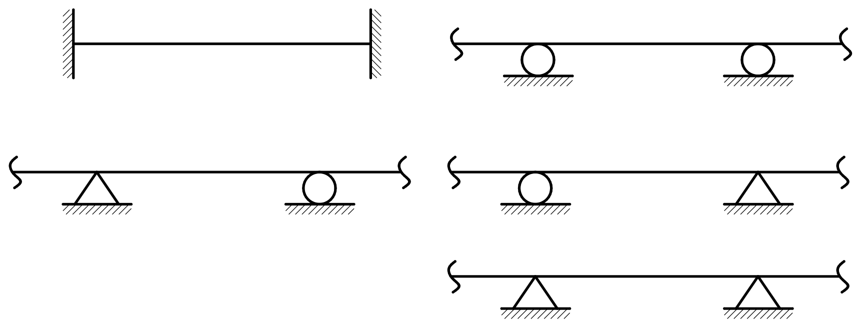
After deciding which value of moment diagram we are going to use for a certain interval of length, the next step is to calculate the area of reinforcement to be inserted into the section. In this section we are going to discuss the procedure of ultimate limit state design of flexural members according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

Flow Chart 16: Design of Beam Section for Flexure

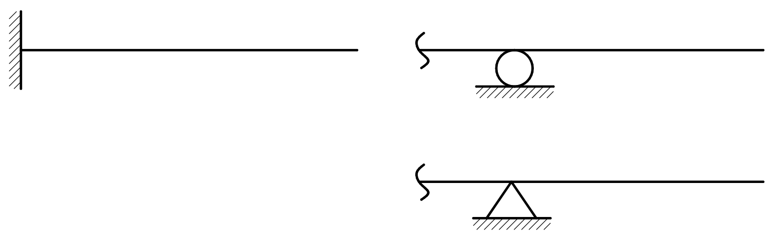




b. Spans considered as end span



c. Spans considered as interior span



d. Spans considered as cantilever

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

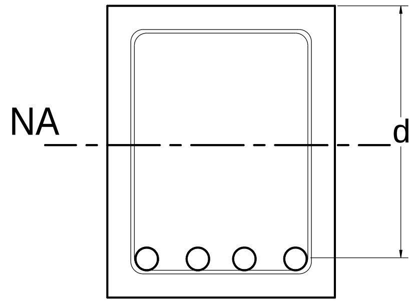


Fig 8-4: Effective depth assumed at the start of design

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.

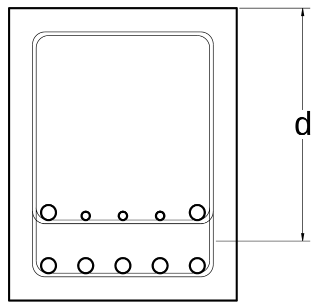


Fig 8-5: Effective depth of a section having more than one layer of bars

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.

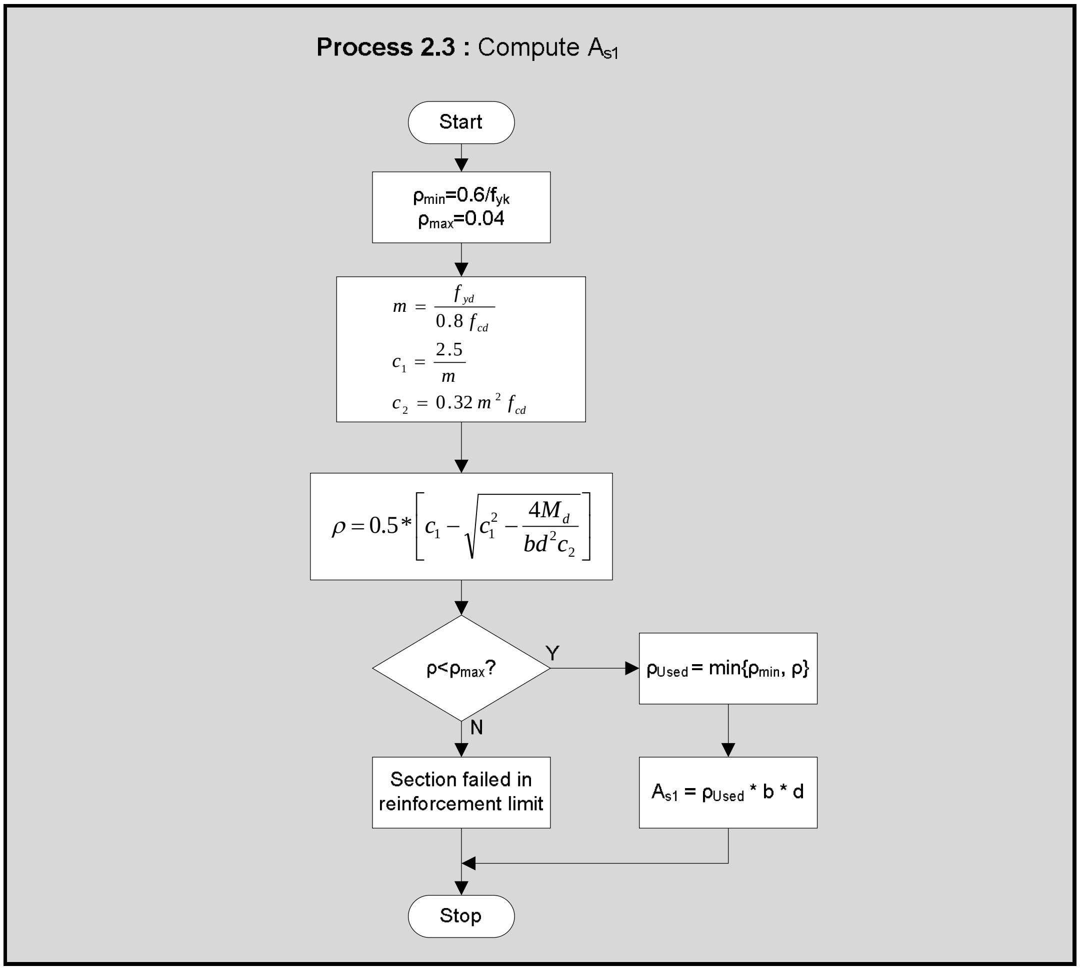
"For continuous beams and for beams in rigid jointed braced frames with span/effective depth ratio not greater than 20."

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

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;

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.

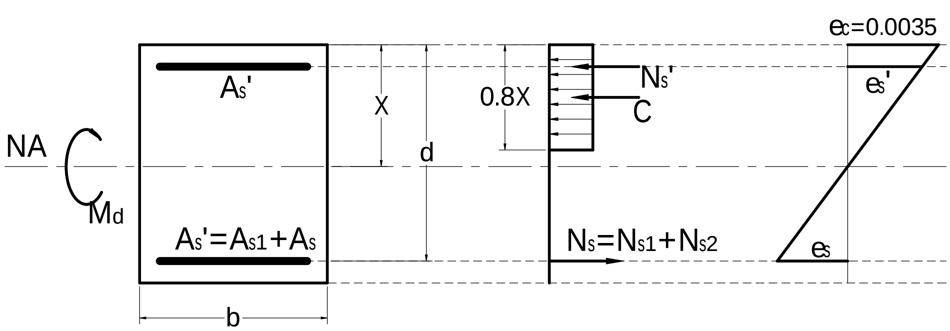
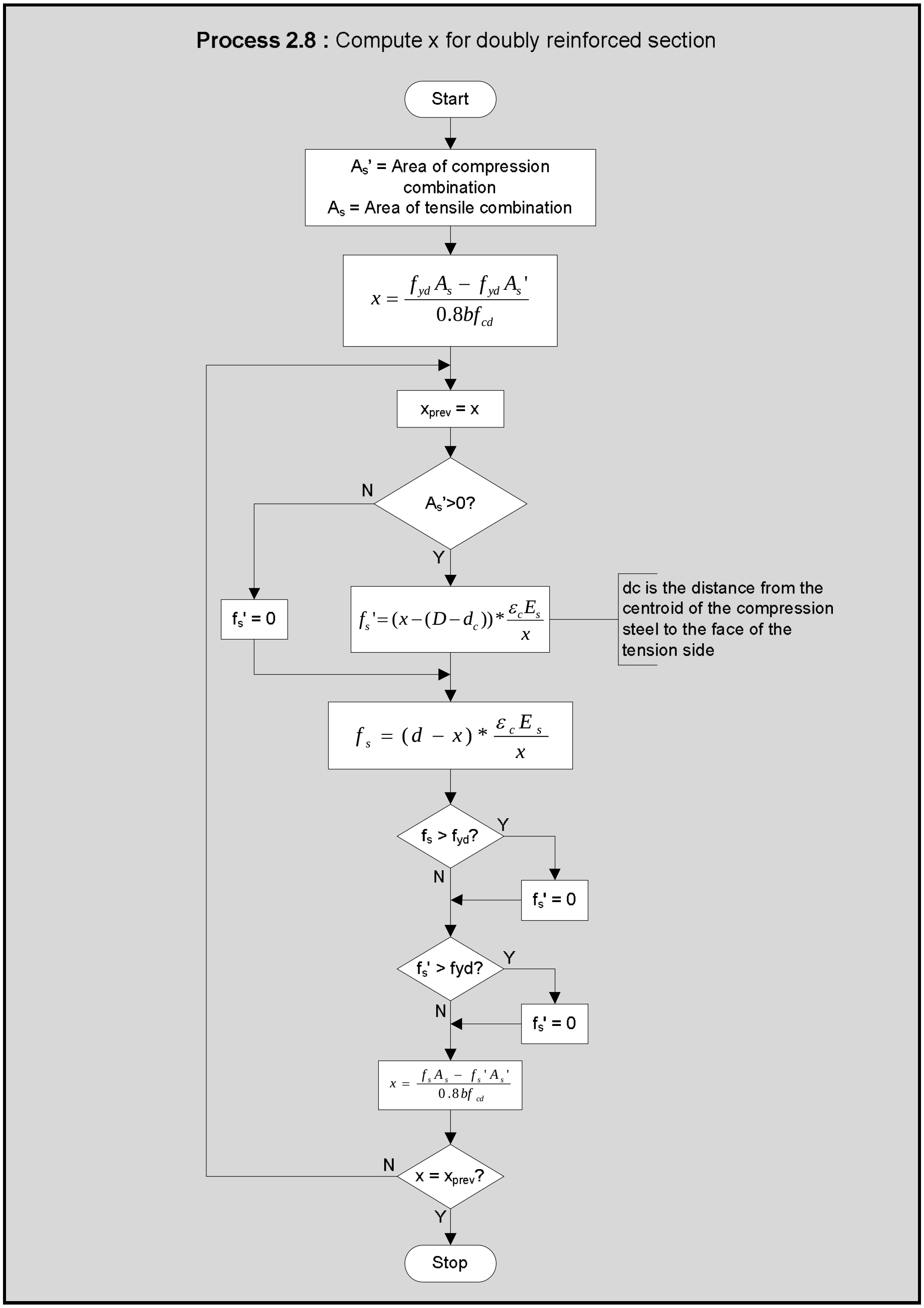


Fig 8-6: Stress strain diagram for doubly reinforced section

$$\sum F_x = 0$$

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.

$$k_x = \rho m$$

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

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

Solving for ρ, we get:

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

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

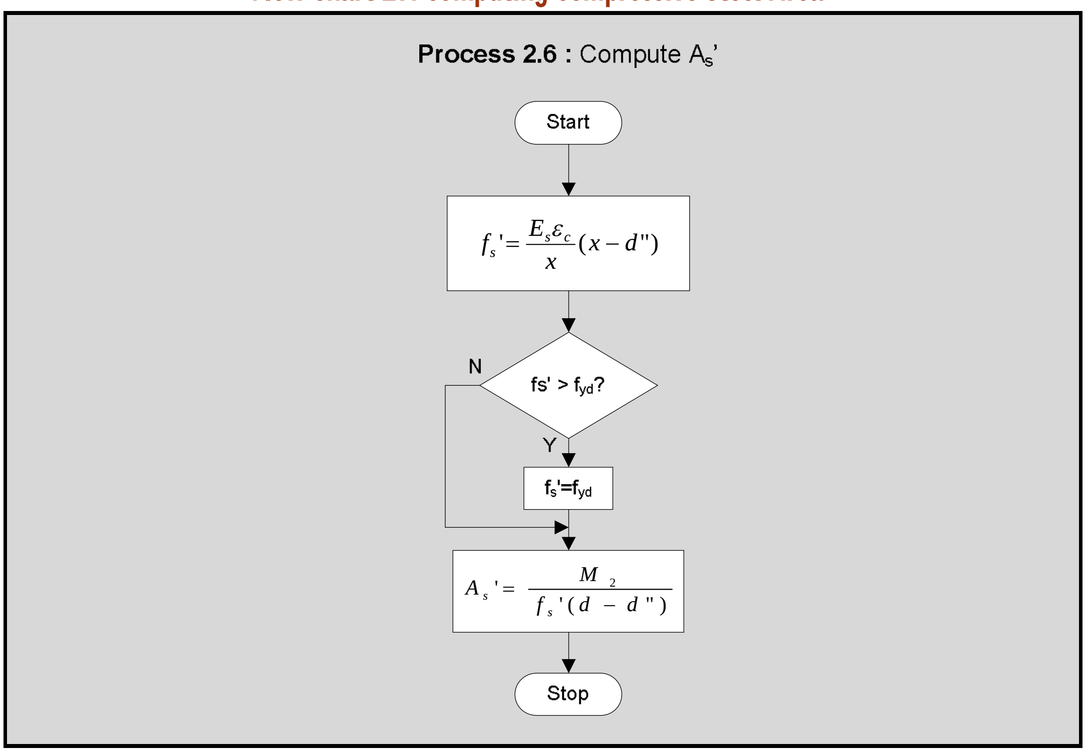
$$\rho_{min} = \frac{0.6}{f_{yd}} \quad \text{8-15}$$

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

Where f_{yk} is in MPa.

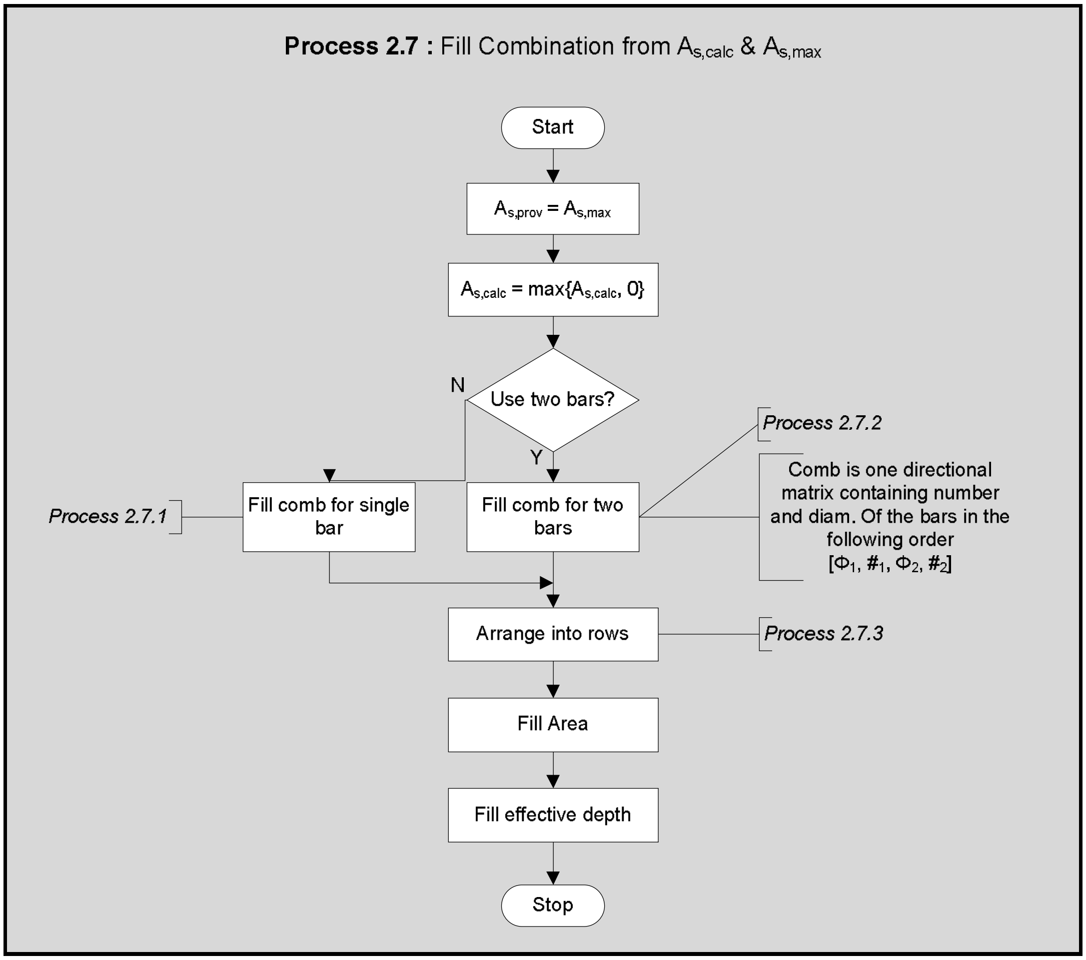
ESADS checks these limits after calculating the steel ratio using the above formula. The formula of the steel ration gives two values of ρ. However, the larger value is never chosen. If the other ratio is greater than ρ_{max}, the section has failed. Otherwise the minimum of ρ_{min} and the calculated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

Flow Chart 27: Computing Compressive Steel Area



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

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.

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

	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.

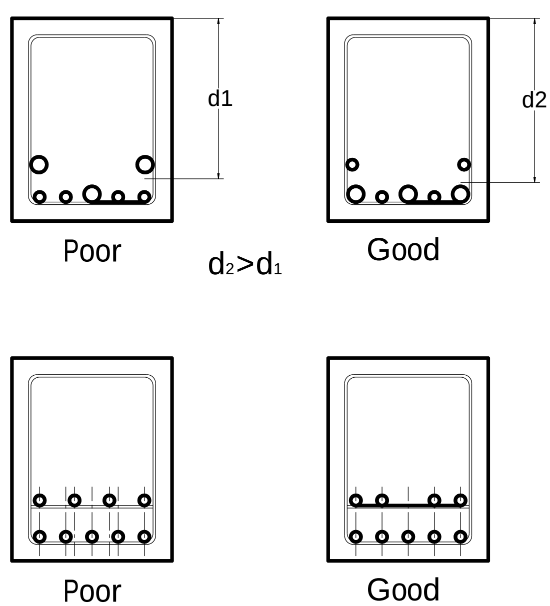


Fig 8-7: Good and poor practices of arrangement of bars in cross-section

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.

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:

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

Where:

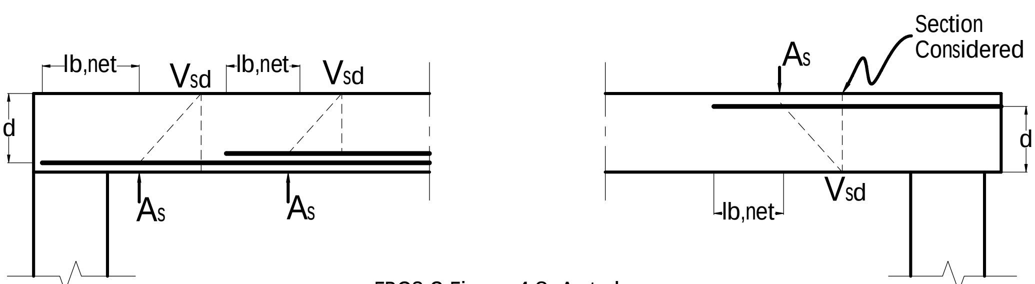
k_1 = $(1+50\rho) \leq 2.0$

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.

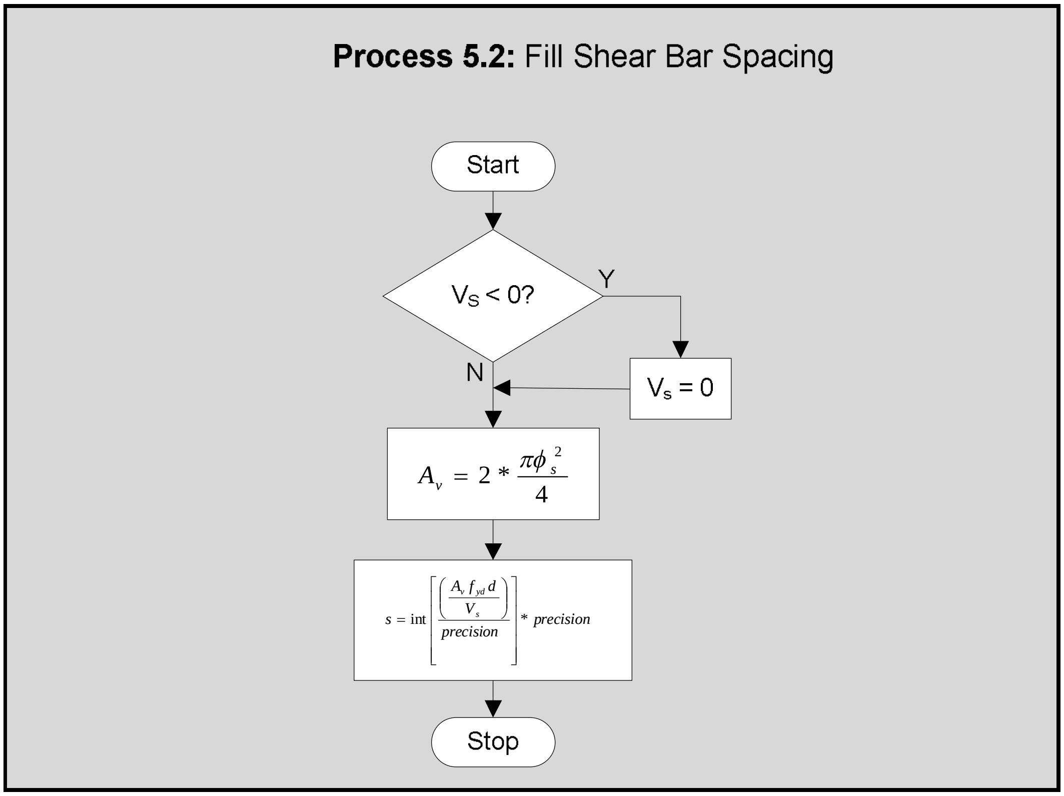
"When shear reinforcement perpendicular to the longitudinal axis is used, its shear resistance, V_S may be calculated as:

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

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

The area, A_V, 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



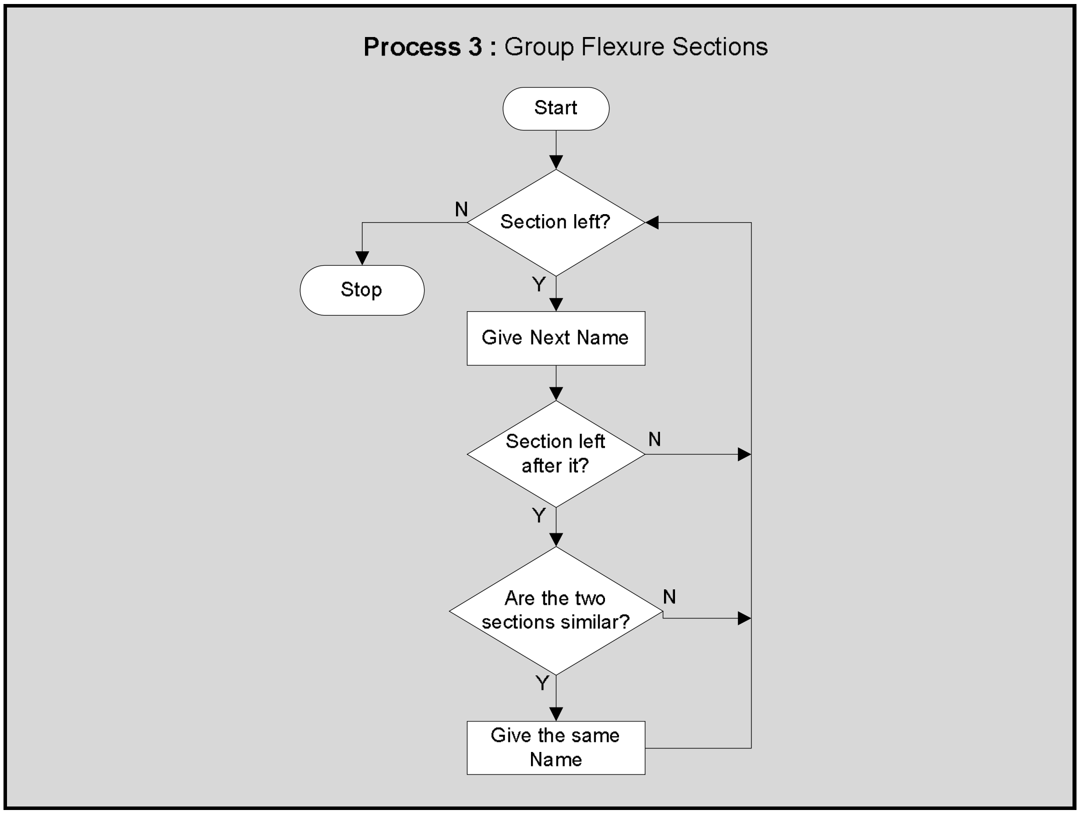
8.4. Design Section Optimization

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

Flow Chart 17: Grouping Sections



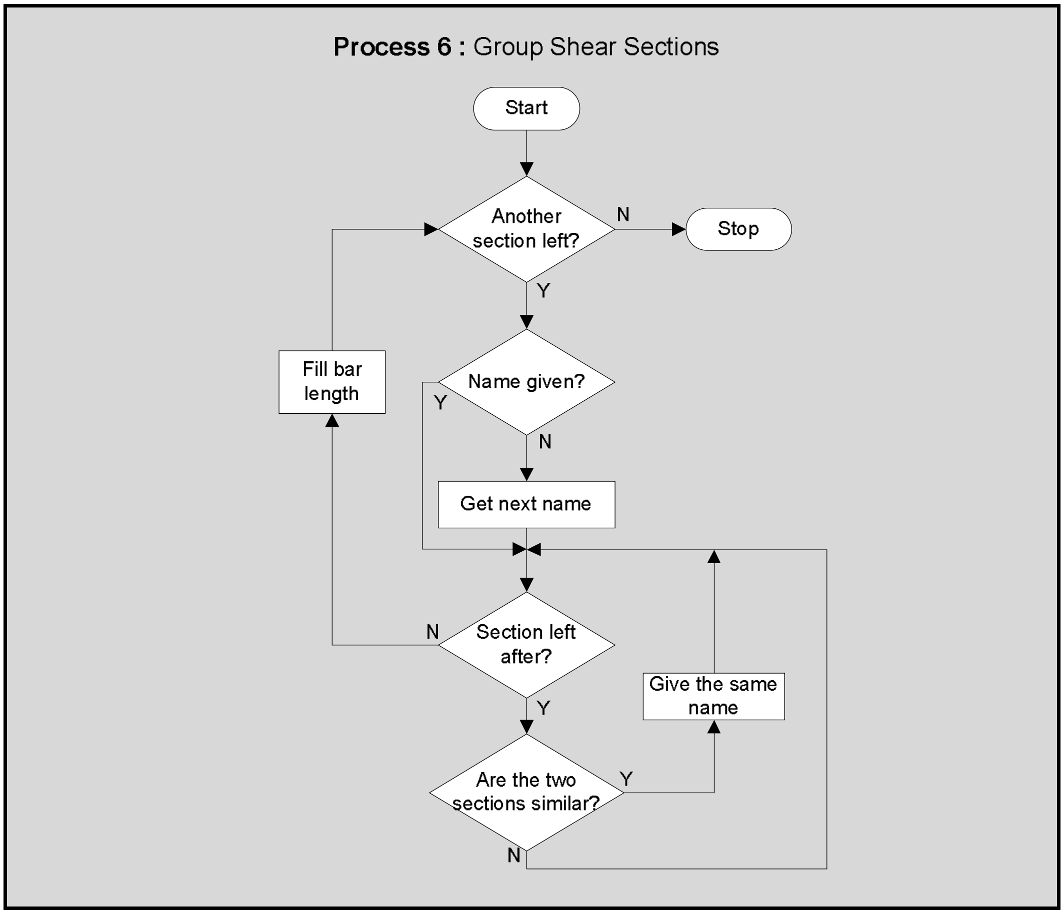
When we come to optimization of shear sections, the designer needs to decide the number of shear sections first. Then an exhaustive list of critical shear values of the shear force diagram, i.e. shear values at d distance from the supports of each member and value of V_C. The relative number of required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

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

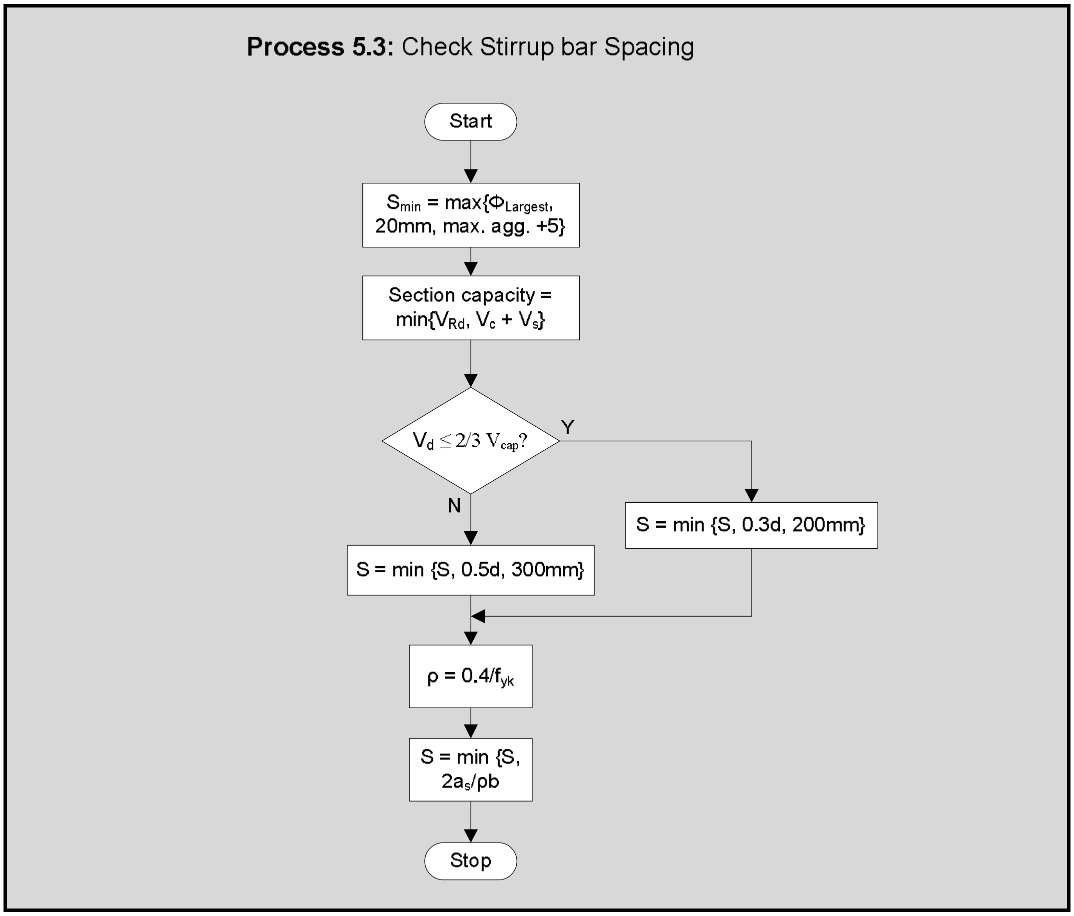
EBCS gives the maximum and minimum values of ρ, the steel ratio:

"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 33: Check Stirrup Bar Spacing



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.

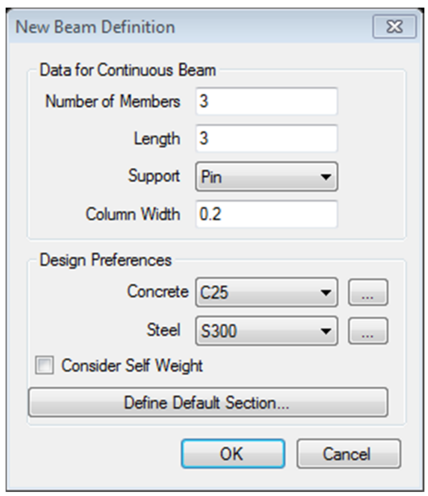


Fig 9-1: New Beam Definition dialog

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.

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.

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.

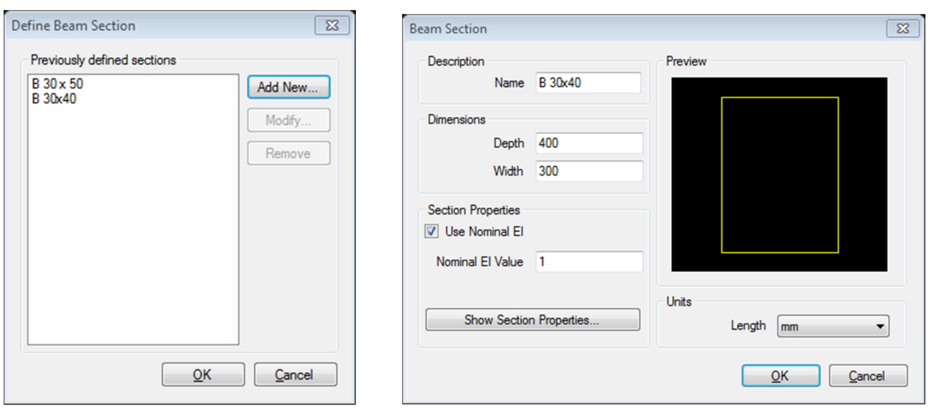


Fig 9-2: Define Beam Section Dialogs

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.

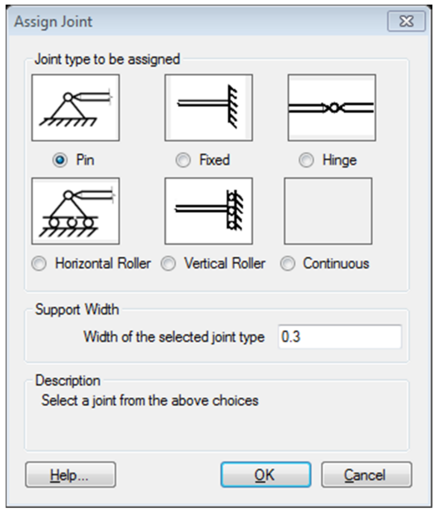


Fig 9-3: Assign Joint Dialog

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:

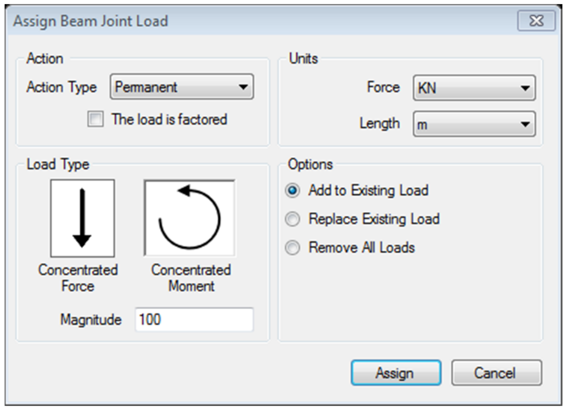


Fig 9-5: Assign Beam Joint Load Dialog

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:

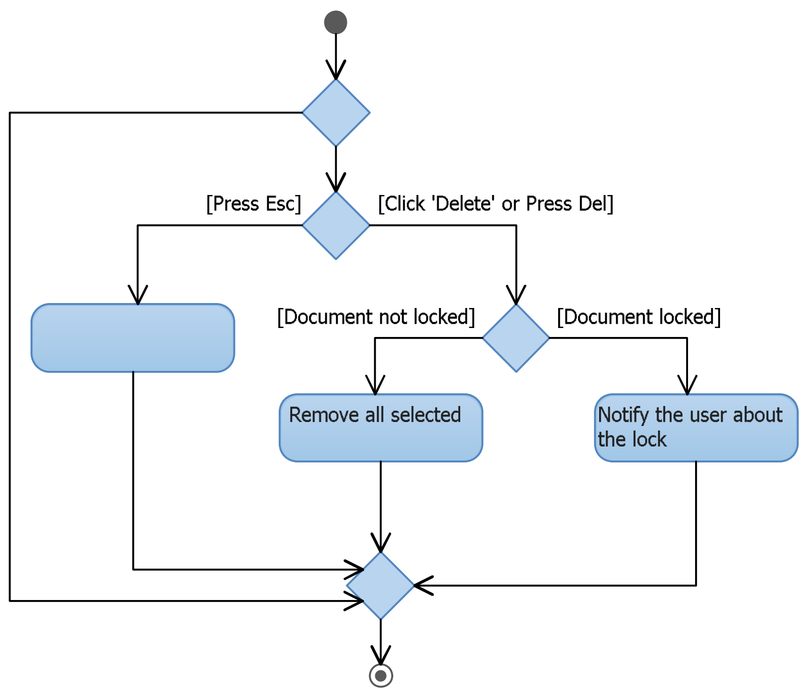
Scenario:

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.

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.

10.2. 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.

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.

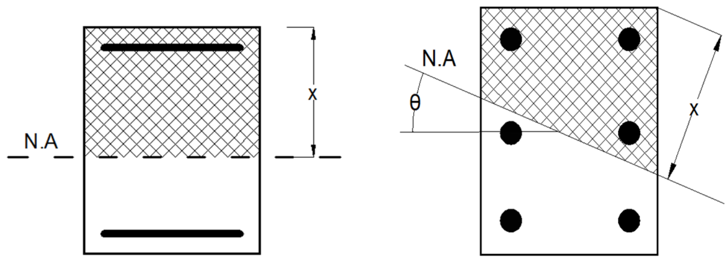


Figure 10-5 Intermediate outputs of uniaxial and biaxial column

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.

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.

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

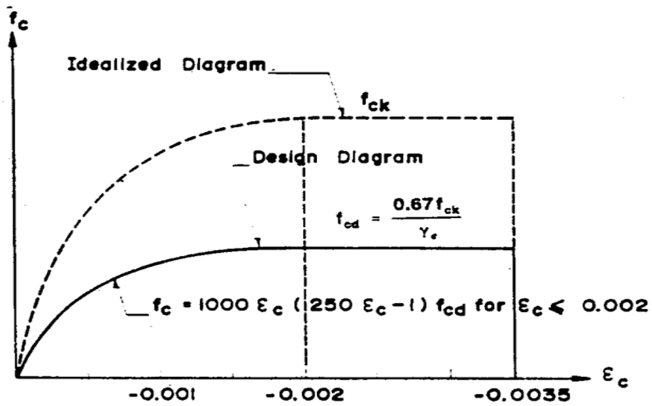


Figure 12-4 Stress strain diagram of concrete

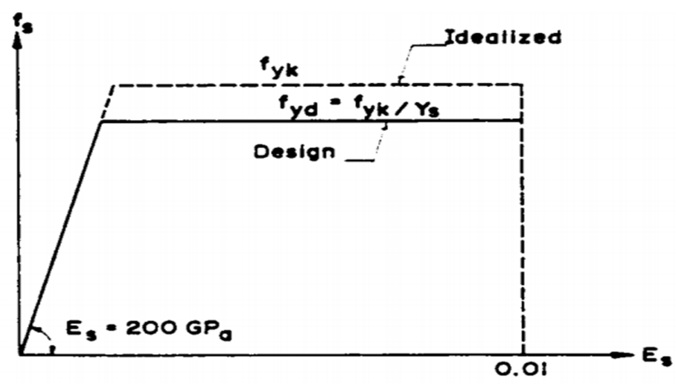


Figure 12-5 Stress strain diagram of steel

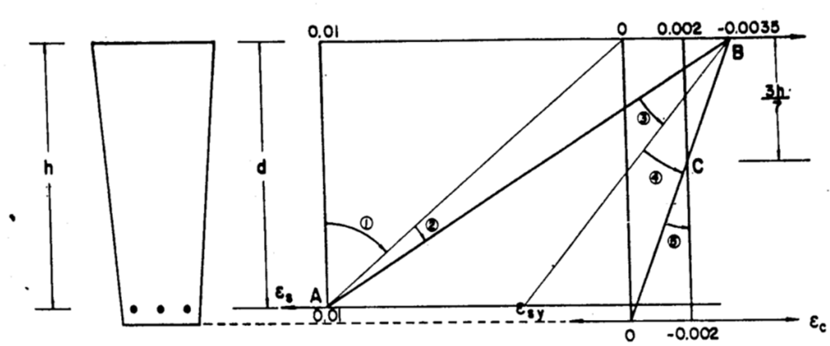


Figure 12-6 Strain diagram at ultimate limit state.

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 compressive strain of concrete εcu = 0.0035, keeping the distance of point C from top compression fiber at $\frac{3}{7}h$.

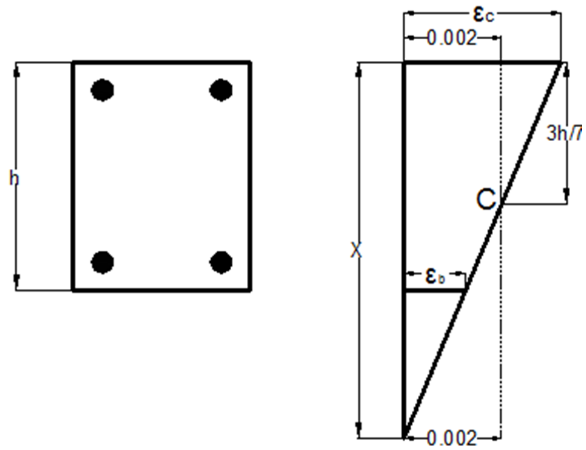


Figure 12-11 Strain in top compression fiber when the neutral axis is out the cross section

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.

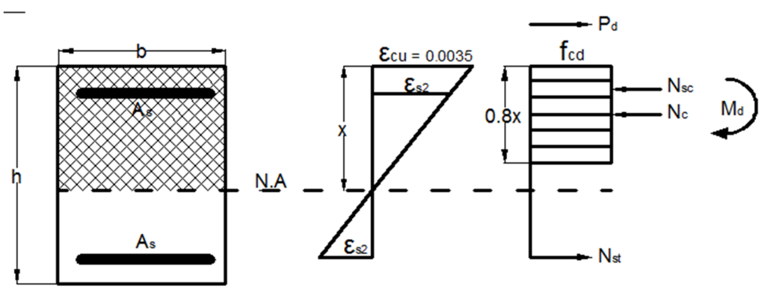


Figure 12-12 General stress strain diagram of uniaxial section with detailing type Type-1

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

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

Taking the summation of moments about the centroid we have.

$$\sum M = 0$$

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

- ✓ 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.

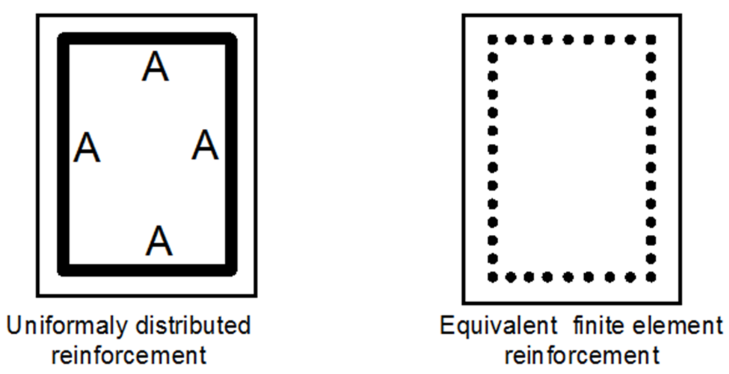


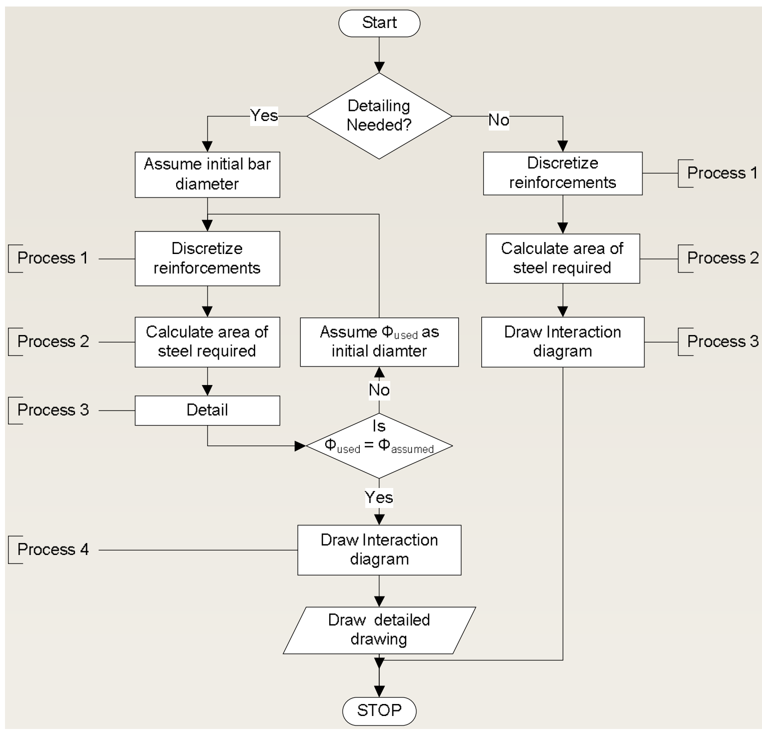
Figure 12-13 Finite element representation of uniformly distributed reinforcement

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

12.4. Design of Uniaxial Column

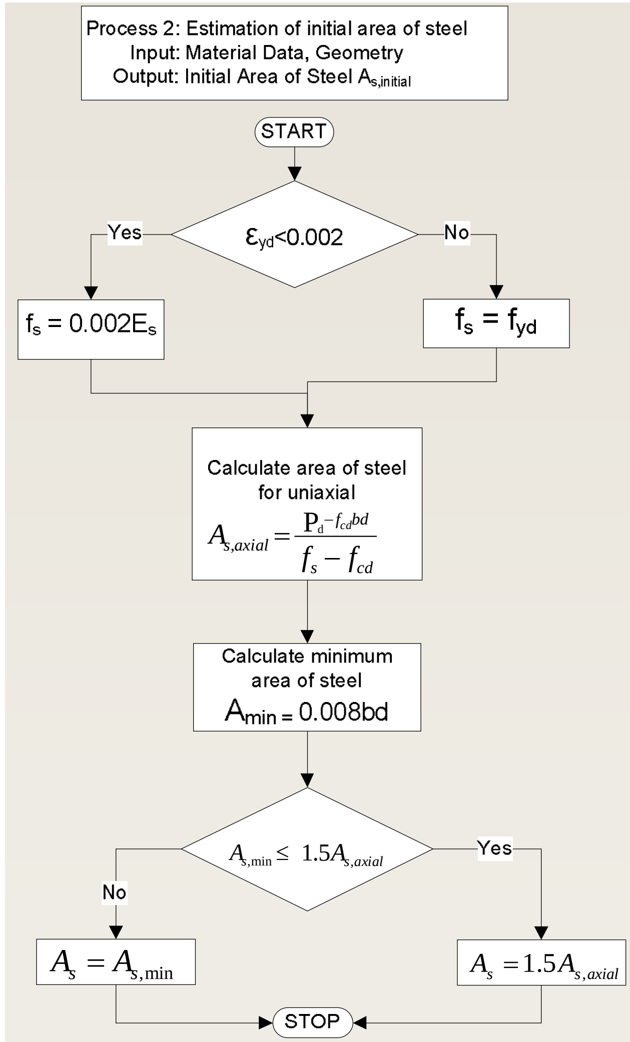
Design of uniaxial column starts by placing of reinforcements with initially assumed area of steel. Then, the section capacity is calculated and checked with the applied design load. If the provided area of steel is insufficient to support the loads, reinforcement area is changed and the design capacity 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.



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.

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.

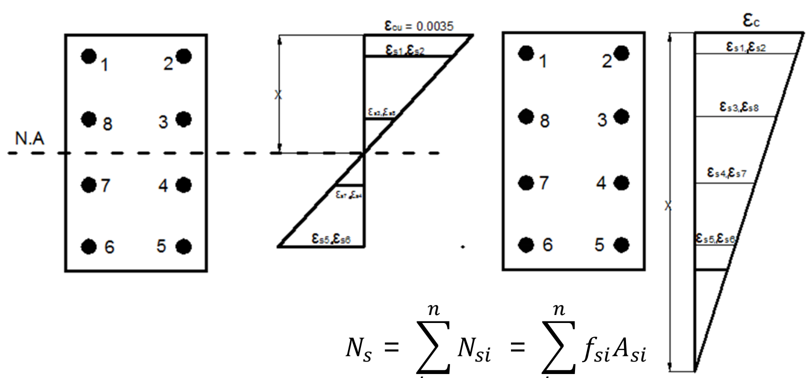
The first step in calculating the force carried by reinforcements is to calculate the strain level of the top compression fiber. While doing so, we have two general cases described in Section 4.2.

When the neutral axis is within the cross section we take $\varepsilon_{cu} = 0.0035$ 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

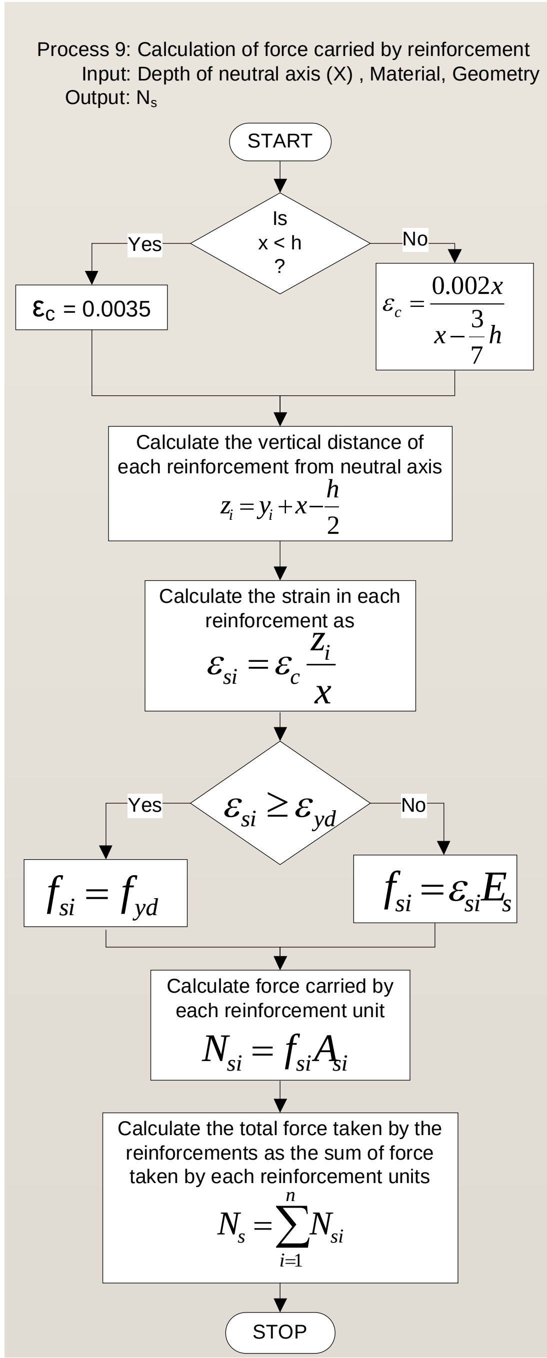


$$N_s = \sum_{i=1}^{n} N_{si} = \sum_{i=1}^{n} f_{si}A_{si}$$

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

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

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



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.

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 $P_{new} \geq P$

$x_2 = x_{new}$

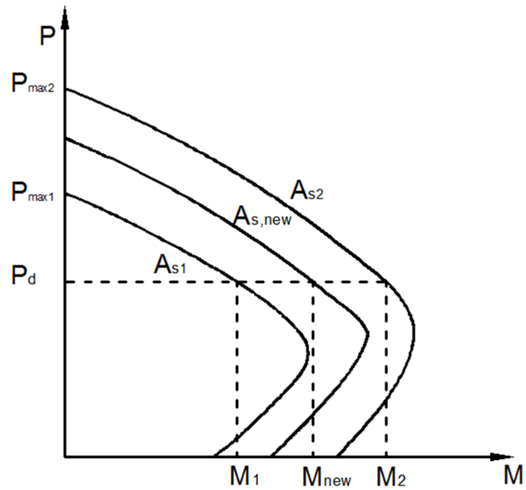
$P_2 = P_{new}$, while $P_1\ and\ x_1$ remain unchanged.

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.

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

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.

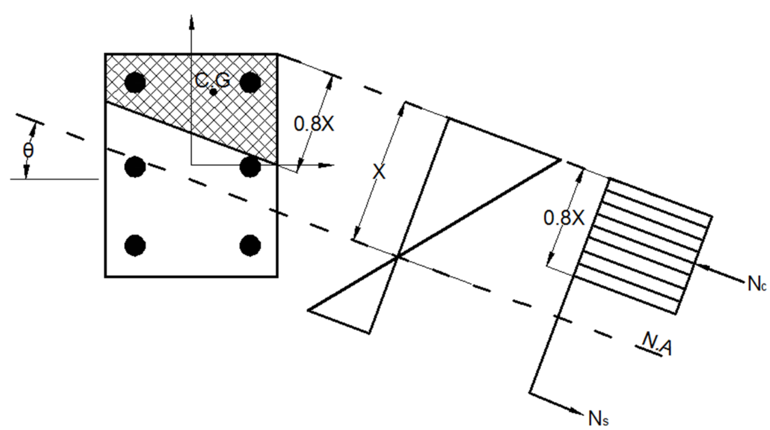
12.5. Design of Biaxial Columns

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

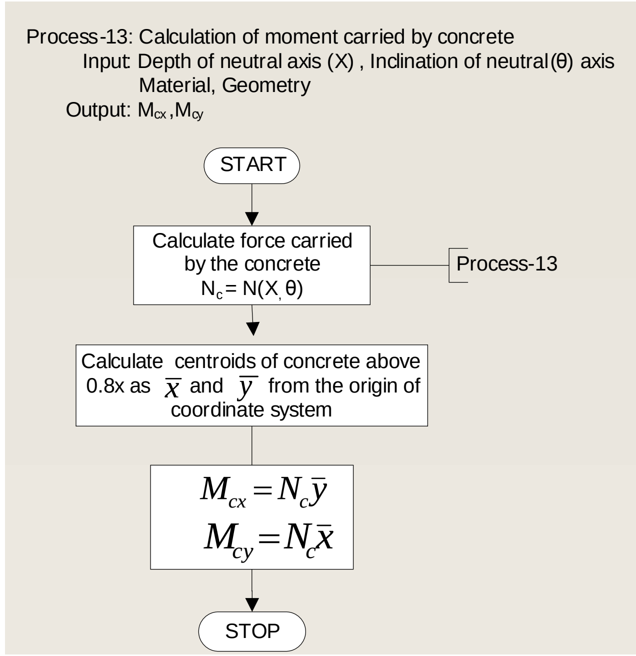
This topic describe the way to calculate the force and moment carried by the concrete for biaxial column for a given depth and inclination of neutral axis. Here we need to compute area and centroid of polygonal compression zone formed as a result of inclined neutral axis. In the same way with uniaxial 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.



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

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

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



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.

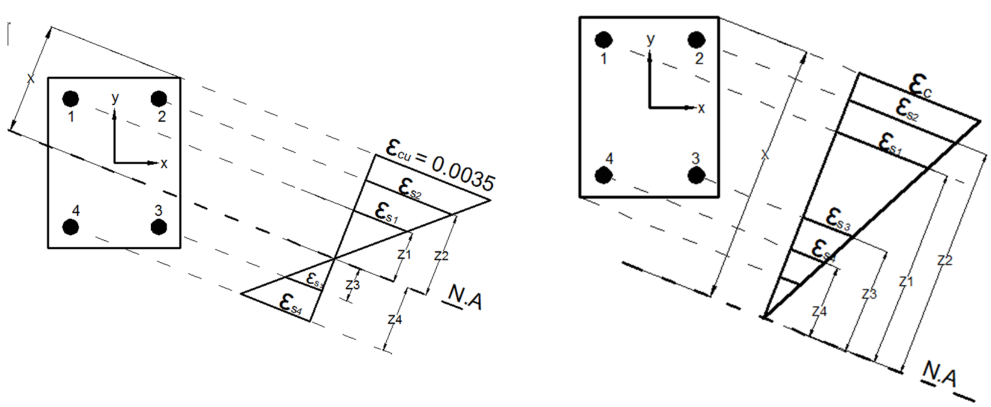
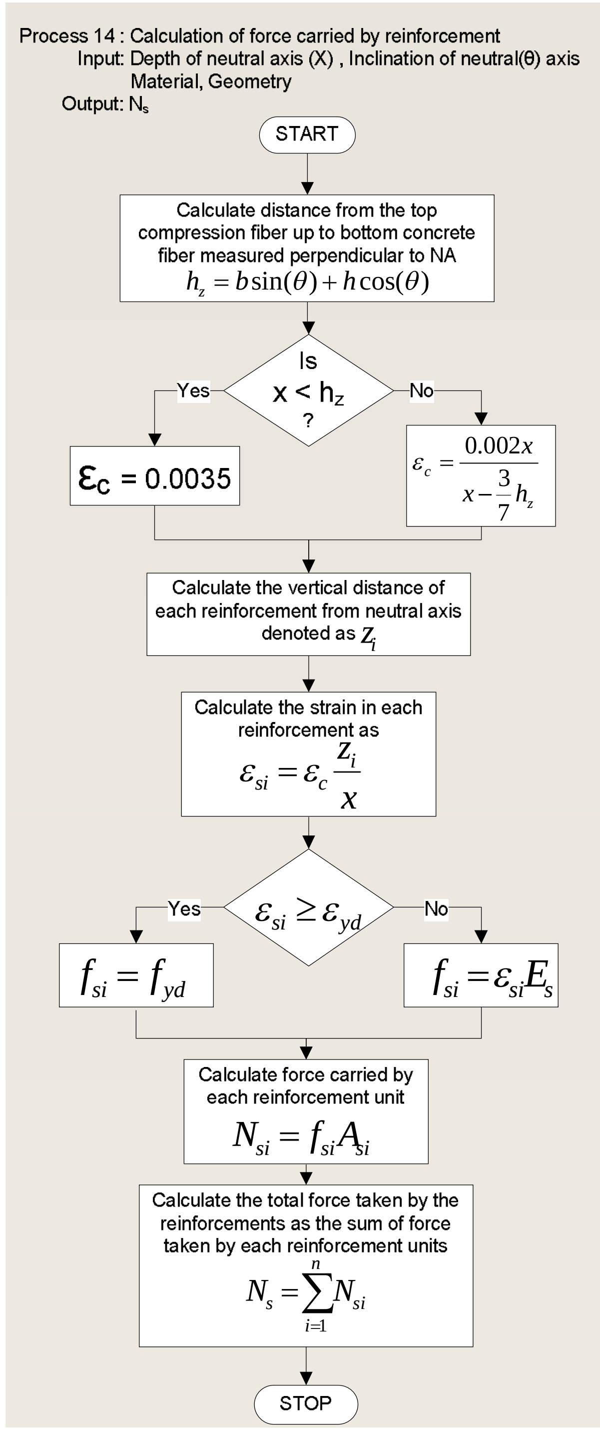


Figure 12-17 Location of neutral axis the left one: when the neutral axis is within the x-section and the right one: when the neutral axis is outside the x-section



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

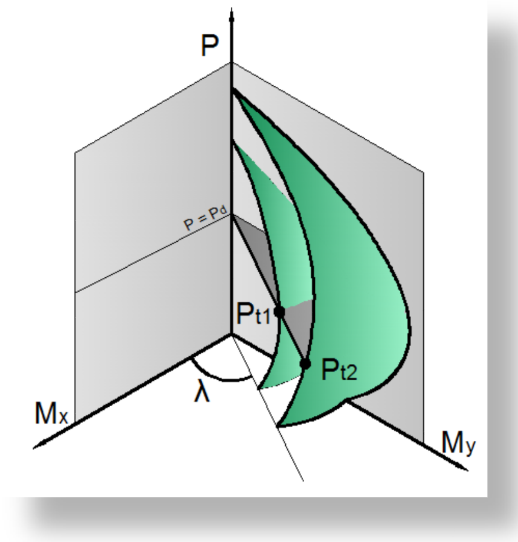


Figure 12-18 Brocken-out section showing interaction surface drawn for two different area of reinforcements.

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,

$A_{s1} = A_{s2}$

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

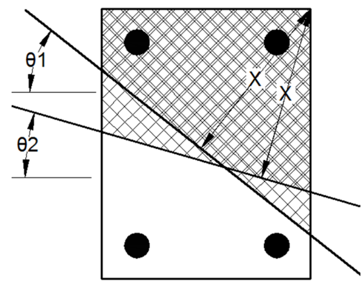
$M_{x1} = M_{x2}$

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

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.

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, $M_x \ and \ M_y$

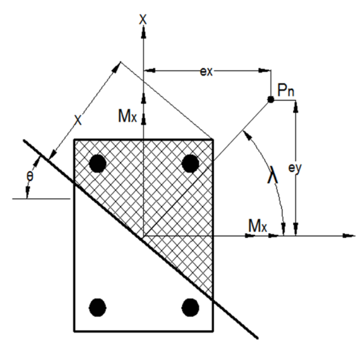
In the design equation derived based on force and moment equilibrium both angle of inclination and depth of neutral axis have contribution. But, the relative effect of x and θ on P, $M_x \ and \ M_y$ is different. For example, if we consider the above figure, changing the angle neutral axis have significant 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.

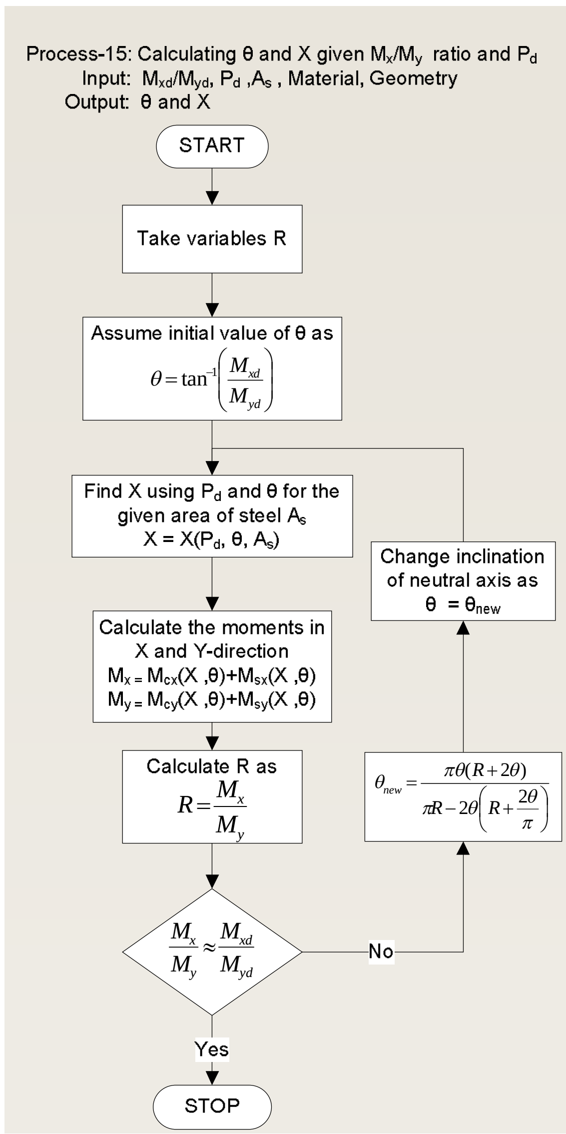
$$M_x = M_{sx}(x, \theta) + M_{sx}(x, \theta)$$

Now 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.

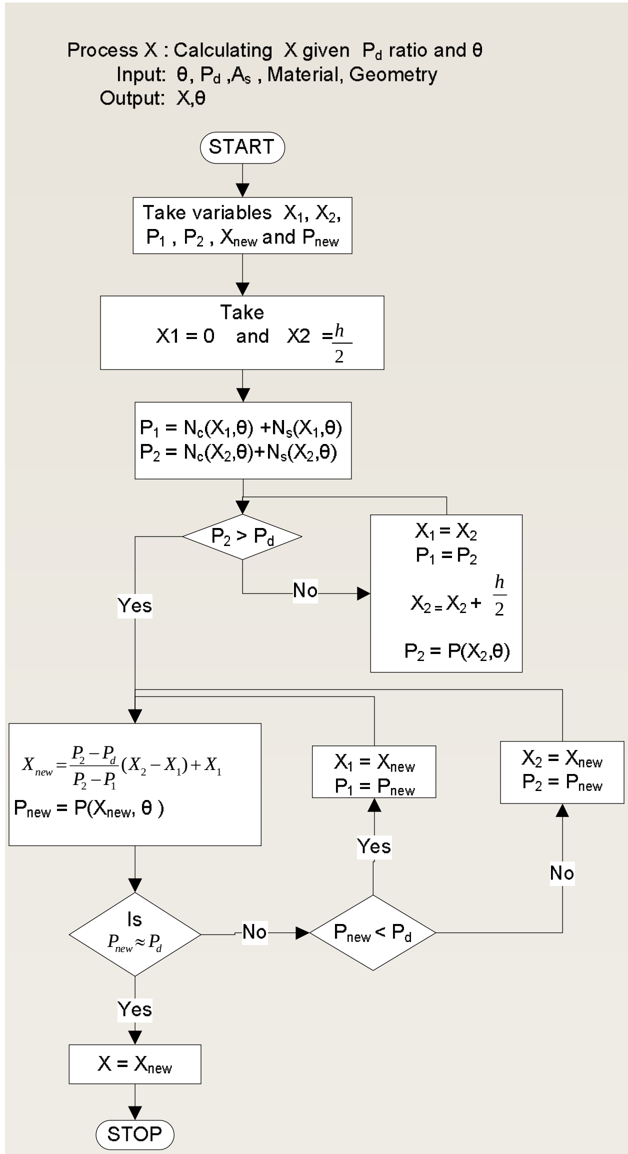
$$M_x = M_x(x, \theta)$$

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

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

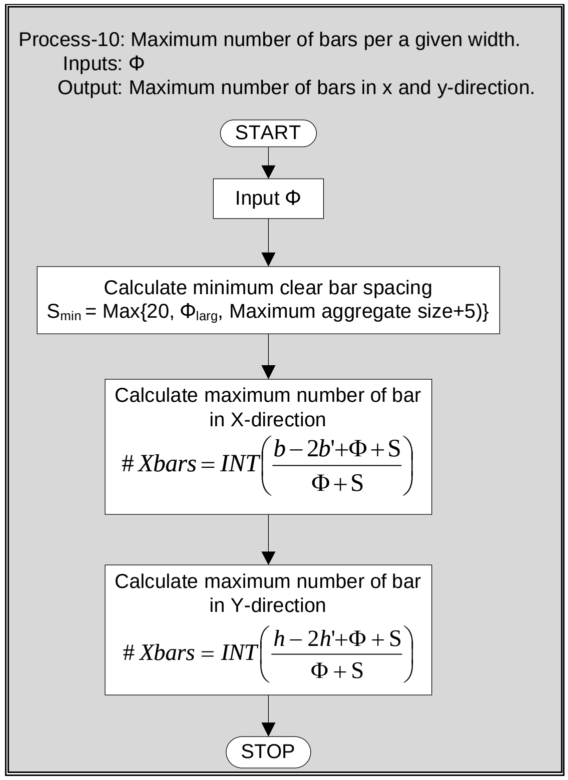


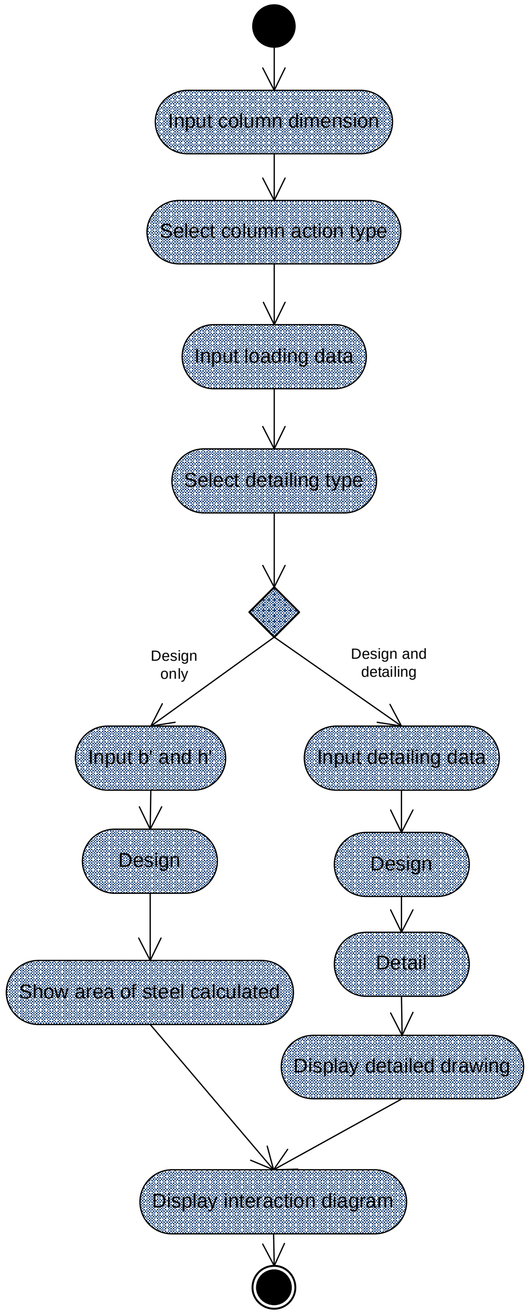
Flow Chart 12-12 Calculation of depth of neutral axis

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,

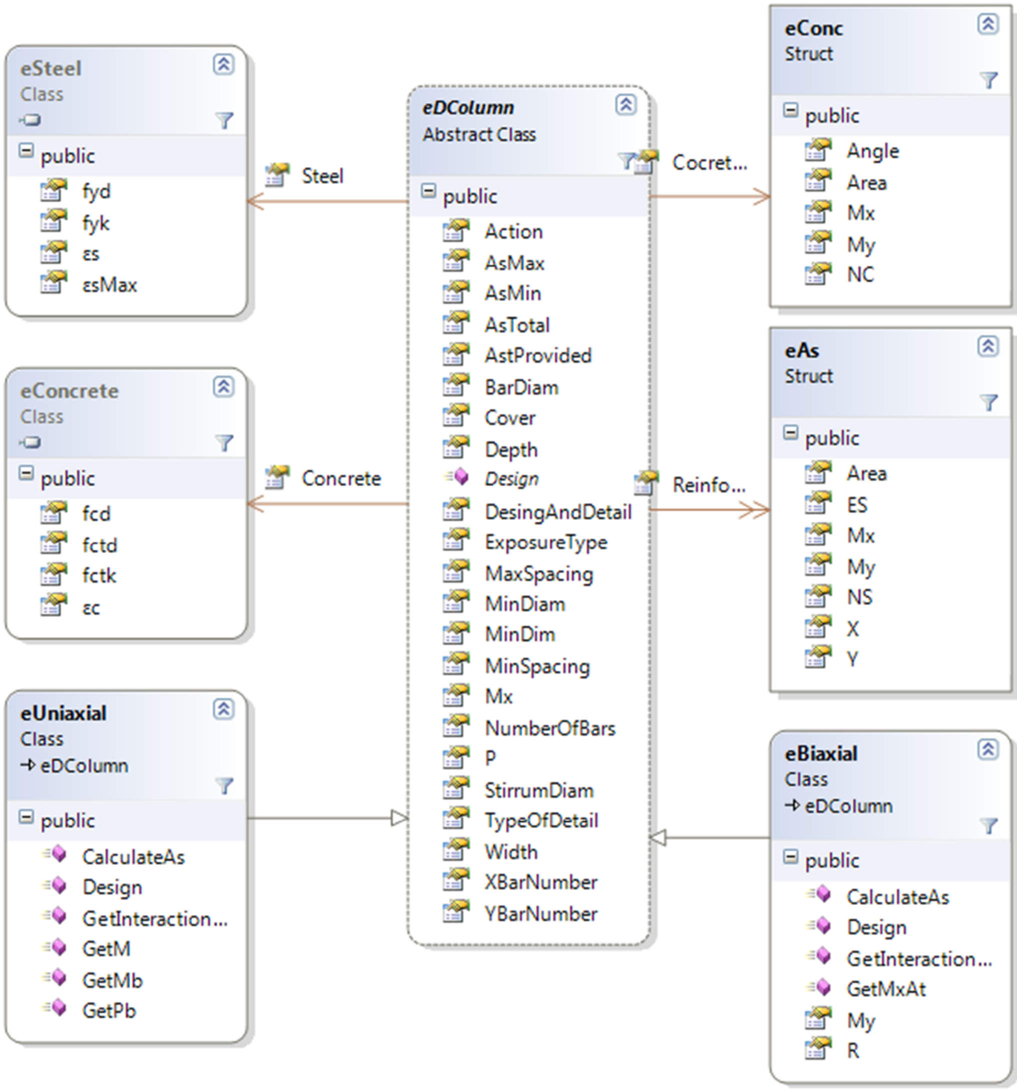
- ✓ Maximum bar diameter
- ✓ Minimum bar diameter
- ✓ Concrete cover or exposure condition
- ✓ Diameter of tie reinforcement
- ✓ Minimum spacing of tie reinforcing
- ✓ Bar diameter, if the user need for the detailing to be done using only one bar


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



13.2. 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).


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
eAs
Struct
public
Area
ES
Mx
My
NS
X
Y
Reinfo...
eConcrete
Class
public
fcd
fctd
fctk
εc
Concrete
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

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 = 1505.24mm²

P_{max} = 3730.4KN

P_b(balanced) = **1287.75**KN

M_b(balanced) = 377.39KNm

M(pure bending) = 216.6KNm

Checking using design charts:

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

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

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

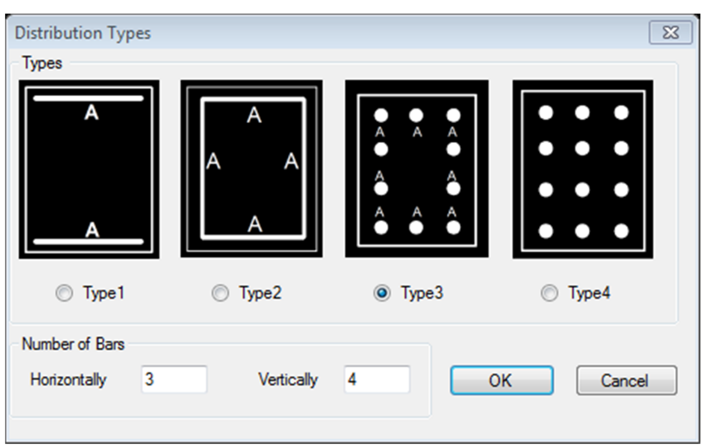
$A_{s,total} = \frac{\omega A_c f_{cd}}{f_{yd}} = 3128.00$

As = 3128.00/2 = 1564.00mm² >1511.01mm² . As we can see, the code value are somewhat conservative.

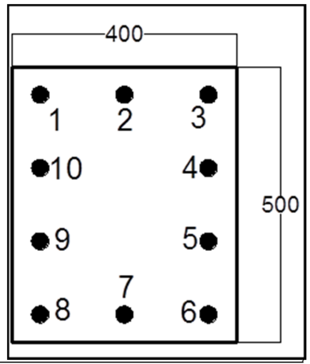
Example2: Uniaxial Column Type3

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

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



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*

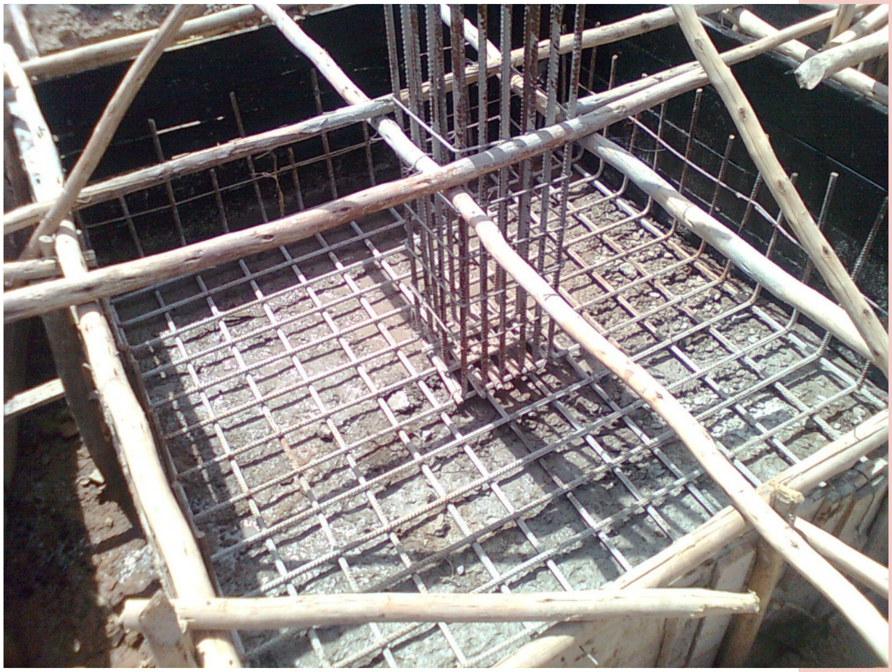
Trail4:

Finally on the last trail we get A = 367.09mm² and X = 357.67mm.

Bar	Area	Y	Z	εs	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 452.39mm² 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 As= 367.087mm², the program calculate the new X to be 357.20mm.

Bar	Area	Y	Z	εs	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***



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.

15. Introduction to Footing

Outline:

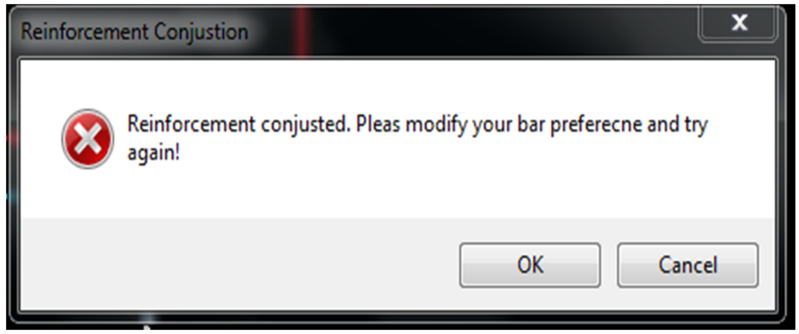
- Overview
- Scope
- Features

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



- ✓ 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”

- ✓ 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.

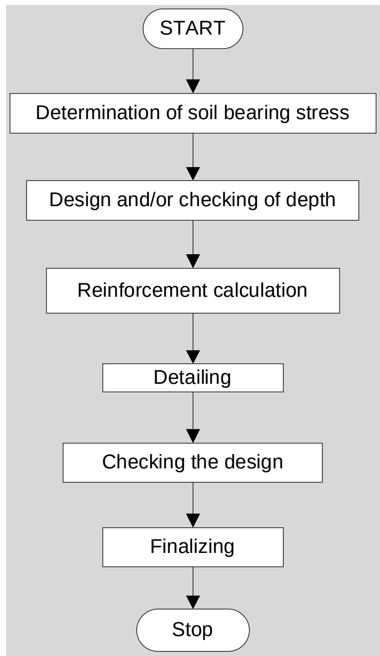
16.3. Manual Method

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.

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.

$$q_u = q_{avg} = \frac{P}{L*B}$$

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)

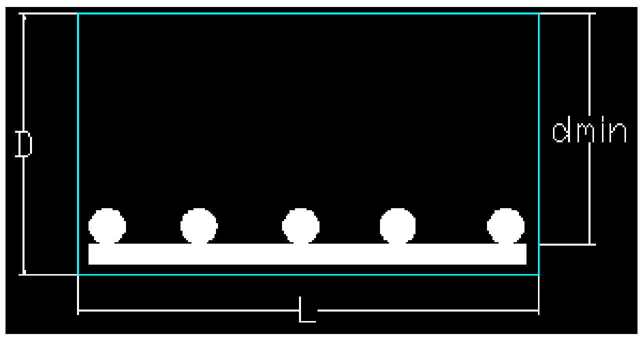


Figure 17-1

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.

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:

$$l_{avail} = \frac{L\ or\ B}{2} - \frac{l_C or b_C}{2} - cover$$

If it doesn't exceeds

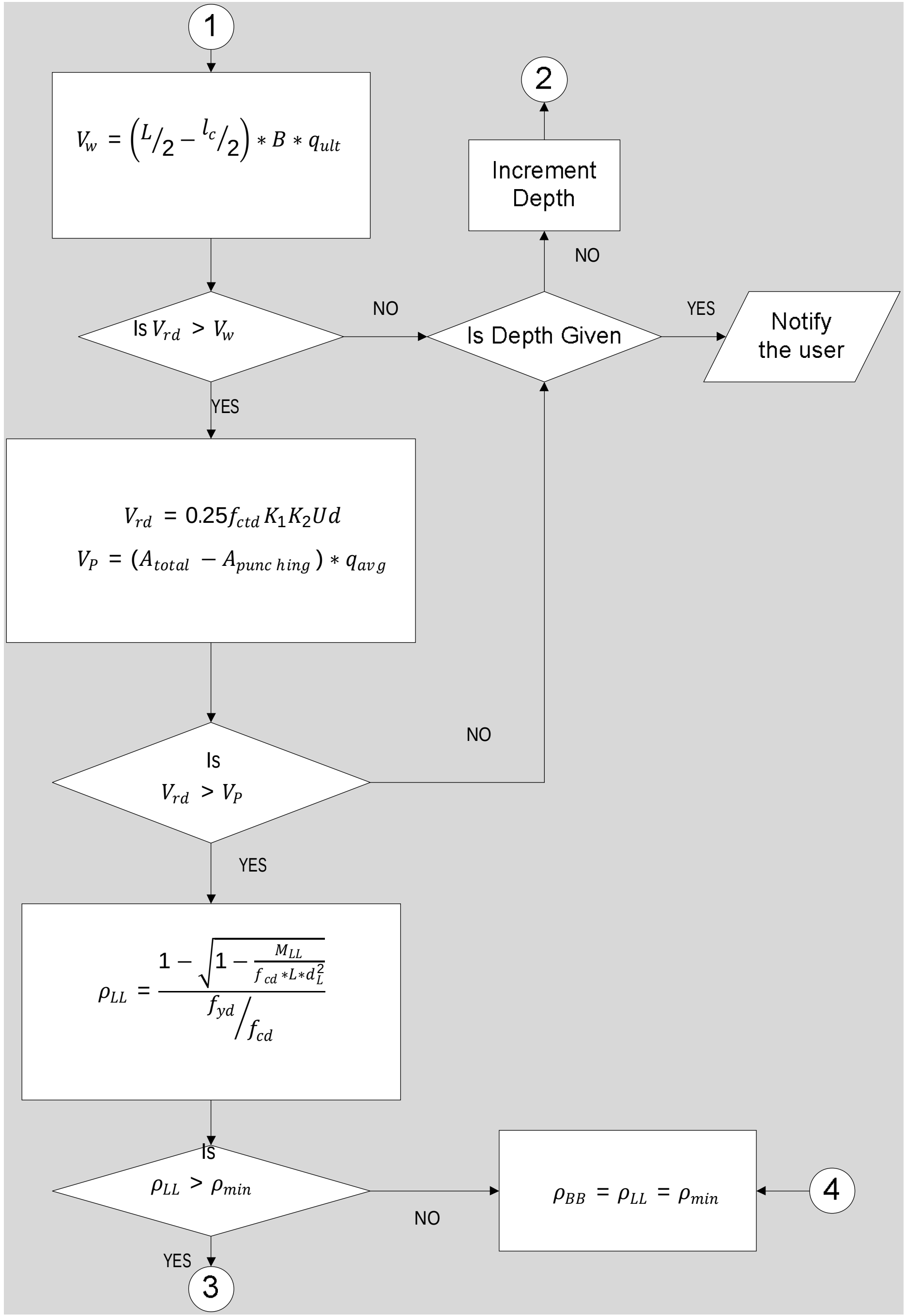
$$l_{avail} = \frac{L\ or\ B}{2} - \frac{l_C or b_C}{2} - d - cover$$

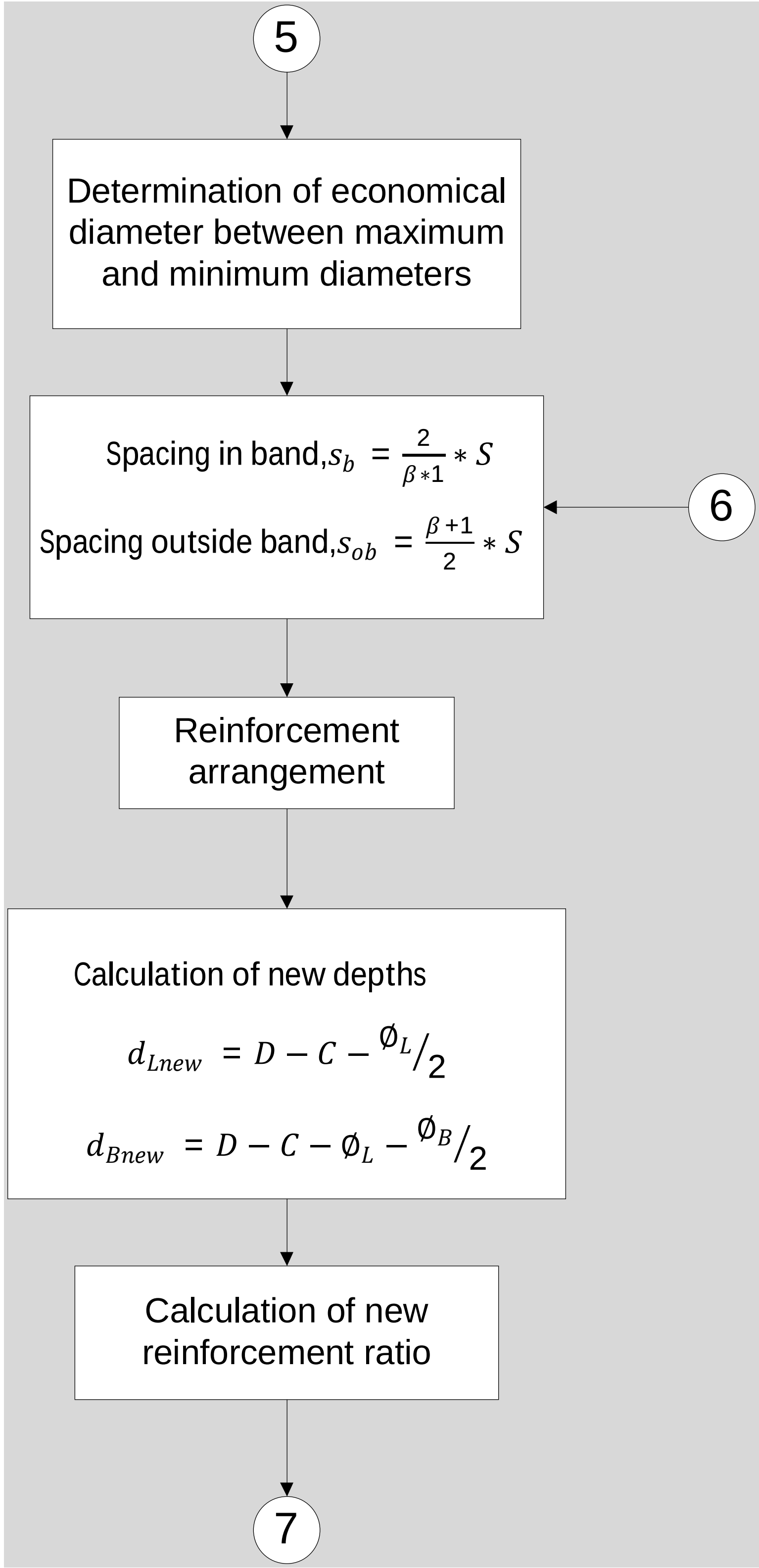
After calculating the available anchorage length it should be checked with net anchorage length and if it is greater we use straight bar otherwise we use hooked bar type. (EBCS sec 7.1.6)

$$l_{bNet} = al_b \frac{A_{scal}}{A_{sprov}} \geq l_{bmin} \qquad l_b = \frac{\emptyset}{4} * \frac{f_{yd}}{f_{bd}} \qquad f_{bd} = 2 *$$
$$f_{ctd}\ \ for\ deformed\ bar$$

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.


1
$V_w = \left(L/2 - l_c/2\right) * B * q_{ult}$
Is $V_{rd} > V_w$
NO
Is Depth Given
YES
Notify the user
NO
Increment Depth
2
YES
$V_{rd} = 0.25 f_{ctd} K_1 K_2 U d$
$V_P = (A_{total} - A_{punching}) * q_{avg}$
Is $V_{rd} > V_P$
NO
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


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

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

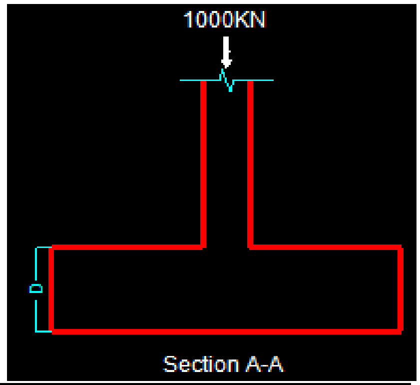
Actor: user

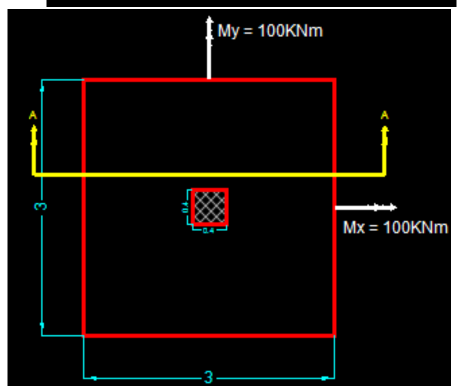
Goal in context: to design an isolated footing

Preconditions: the user must insert all the necessary data to start the design

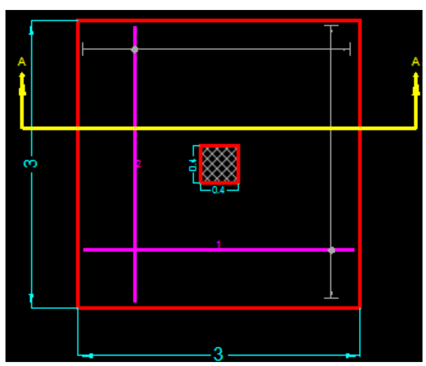
Scenario:

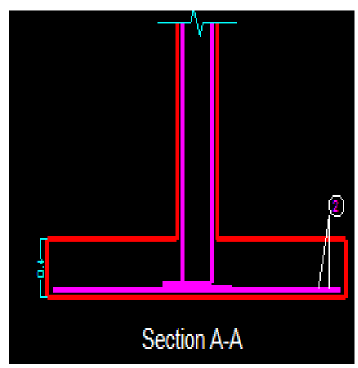
1. From file menu the user selects new model
2. The new model dialog box will appear on the ESADS window and then from those components listed on than dialog box, the user can select footing by clicking on the diagram of footing.

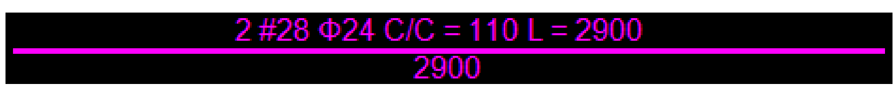


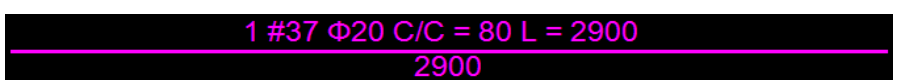


6. From the Run menu the user can run the design by selecting "Run design" option or just by clicking on the short cut "F6".If the data inserted by the user are sufficient enough for the design the user can see the design output which displays in a speed of someone takes to blink his eyes.









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.

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

Appendix