PUBLICATION NUMBER P148

Modelling of Steel Structures for Computer Analysis

Published by:
The Steel Construction Institute
Silwood Park
Ascot
Berkshire SL5 7QN

Telephone: 01344 23345
Fax: 01344 22944

Publication Number: P148

ISBN 1 85942 025 7

British Library Cataloguing-in-Publication Data.
A catalogue record for this book is available from the British Library.

FOREWORD

This guide has been produced as part of the Eureka 130 CIMsteel project, and is the result of collaboration between members of the following organisations:

Ove Arup and Partners
CSC (UK) Limited
QSE
The Steel Construction Institute.

Principal authors of the document are:

Dr David Brohn (QSE)
David Brown (SCI)
Dr Richard Henderson (Ove Arup and Partners)
Alan Rathbone (CSC (UK) Limited)

Valuable comment was received from:

Dr K F Chung (SCI)
David Cunliffe (J N Rowen Limited)
Charles King (SCI)
Dr Mark Lawson (SCI)
Professor David Nethercot (University of Nottingham)
Alan Pottage (Ward Structures Limited)
Peter Purvey (Taywood Engineering Limited)

The guide is primarily aimed at structural designers using readily available analysis software. It is recognised that the use of a powerful program can be counter-productive unless approached with an understanding of real structural behaviour and practical construction. This guide, therefore, offers advice on the modelling of common steel structures for analysis by computer.

CONTENTS

SUMMARY

This document gives guidance on the creation of computer models for steel structures with orthodox details and connections in order to produce safe, cost effective, real structures. It is primarily aimed at structural engineers using readily available analysis software. It highlights the importance of a qualitative understanding of structural response both during the creation of the analysis model and whilst appraising the analysis output.

After a general introduction to the elastic, plastic and elastic-plastic analysis of two and three dimensional frames, separate chapters address the modelling of:

- simple beam and column frames
- trusses and lattice girders
- portal frames
- curved, tapered and non homogeneous members
- connections
- supports
- loads

It also provides guidance on simple checks to ensure the analysis is correct and an overview of member design for the less experienced designer.

This guide is limited to the modelling of general building and plant structures of normal proportions under static loading. Offshore structures, masts, bridges, shells and plates are not covered, nor is grillage analysis. The guide concentrates on first order analysis programs. Second order analysis is discussed, but both the analysis and the type of structure requiring second order analysis are outside the scope of this document.

Modelisation des structures en acier pour une analyse par ordinateur

Résumé

Ce document apporte une guidance pour la réalistion de modèles informatiques de structures en acier, utilisant des détails d'assemblages classiques, afin d'obtenir un dimensionnement sécuritaire, économique et réaliste. Il est principalement destiné aux ingénieurs de structures utilisant des logiciels de structures existants. Il met en évidence l'importance d'une bonne compréhension qualitative du comportement de la structure, tant lors de l'élaboration du modèle informatique que lors de la vérification des résultats de l'analyse.

Après une introduction générale consacrée à l'analyse élastique, plastique et élasto-plastique d'ossatures planes ou à trois dimensions, différents chapitres sont consacrés à la modélisation:

- *des poutres simples et des poteaux des cadres*
- *des treillis et des poutres en treillis*
- *des portiques*
- *des éléments courbes, à hauteur variable ou mixtes*
- *des assemblages*
- *des appuis*
- *des charges*

Il donne aussi des informations quant aux vérifications simples qui permettent de vérifier que l'analyse donne des résultats corrects ainsi qu'un résumé consacré au dimensionnement des éléments, destiné aux ingénieurs peu expérimentés.

Ce guide est limité à la modélisation de structures classiques de bâtiments de dimensions habituelles, soumis à un chargement statique. Les structures minces, les pylônes, les ponts, les coques et plaques ainsi que les grillages de poutres ne sont pas pris en considération. Le guide est surtout consacré aux programmes d'analyse du premier ordre. Les analyses du second ordre sont brièvement discutées mais, tant cette analyse que l'étude des structures qui nécessitent une analyse du second ordre dépassent le cadre de ce guide.

Modellieren von stahlbauten für die computer-berechnung

Zusammenfassung

Diese Publikation gibt Anleitungen zum Entwurf von Computer-Modellen für Stahlbauten mit gewöhnlichen Details und Verbindungen, um sichere, wirtschaftliche, reale Tragwerke herzustellen. Sie ist vorwiegend an Tragwerksplaner gerichtet, die entsprechende Software einsetzen. Sie unterstreicht die Bedeutung, die Antwort des Tragwerks qualitativ zu verstehen, sowohl beim Entwurf des Berechnungsmodells als auch bei der Beurteilung der Ergebnisse.

Nach einer allgemeinen Einführung in die elastische, plastische und elastisch-plastische Berechnung von zwei- und drei-dimesionalen Tragwerken, befassen sich separate Kapitel mit der Modellierung von:

- *einfachen Trägern und Stützen*
- *Raumtragwerken und Fachwerkträgern*
- *Rahmen*
- *gekrümmten, gevouteten und inhomogenen Bauteilen*
- *Verbindungen*
- *Auflagern*
- *Belastungen*

Sie gibt Anleitungen für einfache Kontrollen der Berechnung und einen Überblick über die Bauteilbemessung für den weniger geübten Ingenieur.

Dieser Leitfaden beschränkt sich auf die Modellierung gewöhnlicher Tragwerke mit normalen Abmessungen unter statischen Lasten. Tragwerke der Bereiche Offshore, Maste, Brücken, Schalen, Platten und Trägerroste werden nicht behandelt. Der Leitfaden konzentriert sich auf die Berechnung nach Theorie Erster Ordnung. Die Theorie Zweiter Ordnung wird angesprochen, aber sowohl die Berechnung als auch die Tragwerksarten, die eine Berechnung nach Theorie Zweiter Ordnung erforderlich machen, sind in diesem Dokument nicht erfaßt.

Modelado de estructuras de acero para analisis numerico por ordenador

Resumen

Este documento suministra ayuda para la creación de modelos por oredenador de estructuras de acero con detalles y conexiones estandar con el objetivo de producir estructuras seguras, baratas y prácticas. Está dirigido, en primera instancia, a ingenieros estructurales que esten usando programas de análisis disponibles en la actualidad. Destaca la importancia de la comprensión cualitativa de la respuesta estructural tanto durante la creación del modelo como durante el análisis de los resultados del cálculo.

Después de una introducción general al análisis elástico, plástico y elasto-plástico de pórticos de dos y tres dimensiones, subsiguientes capítulos tratan sobre el modelado de:

- *Vigas y columnas simples*
- *Celosías*
- *Pórticos*
- *Elementos curvos, cuñas y elementos no homogéneos*
- *Conexiones*
- *Apoyos*
- *Cargas*

También se dan ejemplos de comprobaciones simples para asegurarse que el análisis es correcto y un panorama sobre el diseño de elementos para los diseñadores con menos experiencia.

Esta guia se limita al modelado de estructuras generales de proporciones normales y bajo carga estática. La estructuras offshore, mástiles, puentes, tanques y chapas no son tratadas. Este documento se concentra en los programas de análisis de primer orden. El cálculo de segundo orden es tratado, pero tanto el análisis como la distinción de las estructuras que requieren este orden superior se encuentran fuera del alcance de este documento.

Modellazione di strutture in acciaio per l'analisi mediante elaboratore

Sommario

Questa pubblicazione fornisce indicazioni per la modellazione al calcolatore di strutture in acciaio di tipo tradizionale con la finalita' di portare al dimensionamento di sistemi convenienti sia dal punto di vista della sicurezza sia sotto il profilo dei costi. L'attenzione e' stata principalmente rivolta agli ingegneri strutturisti che usano programmi di calcolo per l'analisi strutturale. Viene sottolineata l'importanza della corretta interpretazione, almeno a livello qualitativo, della risposta strutturale agendo sia a livello di creazione del modello di analisi sia nella fase di esame dei risultati finali.

Dopo un'introduzione generale ai diversi tipi di analisi (elastica, plastica e elasto-plastica) per sistemi intelaiati bidimensionali e tridimensionali, in alcuni capitoli viene e' trattata la modellazione di:

- *travi in semplice appoggio e colonne di telaio*
- *elementi biella e travi reticolari*
- *portali*
- *elementi curvi, rastremati e non omogenei*
- *collegamenti*
- *vincoli di appoggio*
- *carichi*

Vengono anche fornite, per progettisti meno esperti, indicazioni di massima per semplici controlli atti a verificare la correttezza dei risultati dell'analisi e delle dimensioni progettate dell'elemento strutturale.

Questa guida e' limitata alla modellazione di edifici comuni e si occupa di strutture con proporzioni in pianta regolari e soggette a carichi statici. Strutture marine, antenne, ponti, cupole e lastre non sono trattate, come pure l'analisi al fuoco. In particolare si concentra l'attenzione sui programmi di analisi del primo ordine. Viene introdotta anche l'analisi del secondo ordine anche se questo tipo di analisi come pure il tipo di struttura che la richiede esula dallo scopo del presente lavoro.

Modellering av stålkonstruktioner för dataanalys

Sammanfattning

Denna publikation ger vägledning om datamodellering av stålkonstruktioner med konventinella förband och detaljer med avsikt att producera säkra ekonomiska konstruktionslösningar. Publikationen är i första hand avsedd för konstruktörer som använder färdigskriven programvara. I publikationen betonas vikten av en grundläggande förståelse för hur konstruktionen är tänkt att fungera både under skapandet av analysmodellen och vid granskandet av analysresultatet.

Efter en allmän introduktion av begreppen elastisk, plastisk och elastisk-plastisk analys av två och tre dimensionella ramar följer kapitel som behandlar modellering av:

- *ramar med icke momenttagande förband*
- *fackverk och ramar*
- *hallramar*
- *krökta, avsmalnande och ickehomogena element*
- *förband*
- *upplag*
- *laster*

Publikationen ger också tips om enkla sätt att kontrollera att analysen är korrekt och en överblick av konstruktion av element (balkar, pelare etc.) för den mindre erfarne konstruktören.

Denna publikation är begränsad till modellering av normala byggnader med rimliga proportioner under statisk last. Offshorekonstruktioner, master, broar, skal och plattor är inte inkluderade, inte heller rustbädds analys. Publikationen är koncentrerad på programvara som behandlar första ordingens effekter. Andra ordningens analys diskuteras, men både andra ordningens analys liksom de byggnader som kan tänkas behöva andra ordningens analys är utanför vad som behandlas i denna publikation.

1 INTRODUCTION

The power of computers and structural analysis programs has increased dramatically over the past twenty years. Now even the smallest design office has access to a personal computer with at least a two or three-dimensional elastic analysis program. Although technology has advanced, there is increasing evidence that analysis programs are being used without an understanding of the actual behaviour of real structures, and with an unrealistic confidence in the analysis results. The fabrication industry reports increasing incidences of designs that are overly complex, resulting in expensive fabrication details and a loss of cost effectiveness. In some cases designs have been presented which do not represent reality, for example with support conditions that will not be achieved in practice, which, in the extreme cases, could invalidate the design.

The Standing Committee on Structural Safety* recognised the possibility that the use of computers for structural engineering calculations may lead to unsafe structures and, in their Tenth Report(1), identified the following opportunities for unsafe design:

- Persons without adequate structural engineering knowledge or training may carry out the structural analysis.
- There may be communication gaps between the design initiator and the computer program writer and user.
- A program may be used out of context.
- The checking process may not be sufficiently fundamental.
- The limitations of the program may not be sufficiently apparent to the user.
- For unusual structures, even experienced engineers may not intuitively appreciate weaknesses in programs for analysis or detailing.

It is also increasingly common for analysis (and then design) to be carried out with rigid connections throughout the structure. This invariably results in complex connection design, extensive local stiffening in connection zones and highly expensive fabrication. Cost comparisons between simple (pinned) and rigid construction were studied in the earlier phase of the CIMsteel project, and the reader is referred to the publication *Design for Manufacture Guidelines*(2).

This document gives guidance on the creation of computer models for steel structures with orthodox details and connections, in order to produce safe, cost effective real structures. It highlights the importance of a qualitative understanding of structural response, both during the creation of the analysis model and whilst appraising the analysis output.

A computer analysis program should be treated as an extremely useful servant, and not allowed to become a bad master. An expensive, extensively stiffened structure that results from a poor design is generally the responsibility of the structural designer, not the program!

---

* The Standing Committee on Structural Safety is an independent body established by the Institutions of Civil and Structural Engineers to maintain a continuing review of building and civil engineering matters affecting safety of structures.

2 SCOPE

This guide is limited to the modelling of general building and plant structures of normal proportions under static loading. Offshore structures, masts, bridges, shells and plates are not covered, nor is grillage analysis. The guide concentrates on first-order analysis programs. Second-order analysis is discussed, but both the analysis and the type of structure requiring second order analysis are outside the scope of this document.

Many analysis programs are linked to complementary design programs, making member design possible with just a few key strokes. This publication includes a section devoted to member design in an attempt to alert the unwary to some of the possible pitfalls.

Within this guide the following distinctions should be noted:

Analysis: The determination of forces and bending moments.

Design: The selection and checking of member sizes.

Element: The link between nodes in the analysis model.

Member: A component of the real structure comprising of one, or more elements.

In this document, references are made to BS 5950: Part 1[(3)]. All references to Clause numbers refer to the Clauses of that Standard, unless otherwise noted.

3 A QUALITATIVE UNDERSTANDING OF STRUCTURAL BEHAVIOUR

The use of computers in the analysis of structures enables the rapid calculation of forces and moments within a complex frame, by the rigorous application of proven theory and mathematics. Analysis by computer offers advantages to the structural designer in speed and in the accuracy of the arithmetic. There is, however, a growing concern that reliance on computer analysis can seriously reduce the structural designer's ability to understand intuitively the real behaviour of a structure. This concern is not new. The following is taken from a 1956 publication by Nervi[4]:

"The pre-eminence given to mathematics in our schools of engineering, a purely analytical basis of the theory of elasticity, and its intrinsic difficulties, persuade the young student that there is limitless potency in theoretical calculations and give him blind faith in their results. Under these conditions neither students nor teachers try to understand and feel intuitively the physical reality of a structure, how it moves under a load, and how the various elements of a statically indeterminate system react among themselves. Today everything is done by theoretical calculation. That student is rated best who best knows how to set up and solve mathematical equations.

It is highly regrettable that some of the highest qualities of the human mind, such as intuition and direct apprehension,... have been overwhelmed by abstract and impersonal mathematical formulas".

The ability of graduates to understand structural behaviour intuitively was researched in 1977[5] and conclusions reached that the general understanding was poor[6]. Since that study, the use of computer analysis has become the norm, and there is no reason to believe that today's undergraduates have any better intuitive understanding of structural behaviour. It is, however, recognised that with the pre-eminence of computer analysis, an intuitive understanding becomes increasingly important, both in the creation of analysis models and critically, in the appraisal of the analysis results.

This intuitive understanding is termed a qualitative approach, and embodies 'understanding', 'appreciation' and 'intuition'. A qualitative approach involves a non-numerical consideration of the structure and its behaviour.

3.1 Understanding structural behaviour

Numerical analysis of structures is built on an understanding of algebra, geometry and calculus. A qualitative approach uses broader, more intuitive and dynamic reasoning skills to evaluate the behaviour of any particular structure. The key principles involved in developing a qualitative understanding of structural behaviour are:

- To consider the deformed shape of a structure.
- To reduce complex structures into statically determinate, simple systems from which the true structure may be rebuilt.

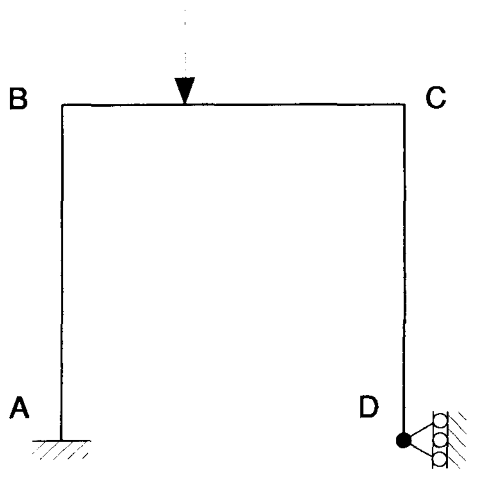


Figure 1 *Typical frame for assessment*

Figure 1 demonstrates the application of a qualitative approach. When asked to draw the approximate bending moment diagram, many recently qualified graduates conclude that the solution is that shown in Figure 2. This should immediately be recognised as incorrect, as the sagging moment under the load can only be sustained if there is a vertical reaction at D.

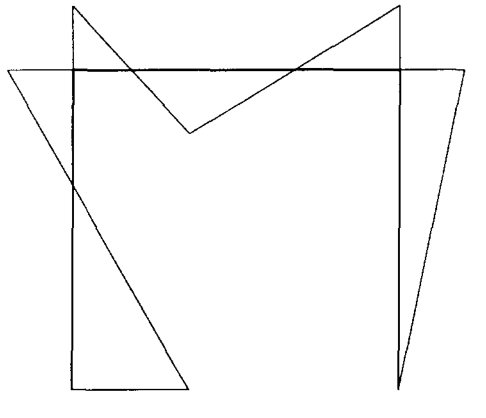


Figure 2 *Common (incorrect) solution to Figure 1*

The correct solution is easily obtained by a qualitative application of the Flexibility Method of analysis. If the obvious release is chosen as the horizontal reaction D, the resulting bending moment diagram for the released structure is shown in Figure 3. The deflected shape, (Figure 4), shows that the reaction at D must be to the right, and the bending moment diagram is shown in Figure 5. The final solution is the combination of both bending moment diagrams, as shown in Figure 6.

The basic rules to a qualitative understanding may be taught relatively quickly, but their application requires practice and improves with experience. A number of frames are included in Appendix A for those who wish to develop (or measure) their understanding by attempting to draw the bending moment diagrams.

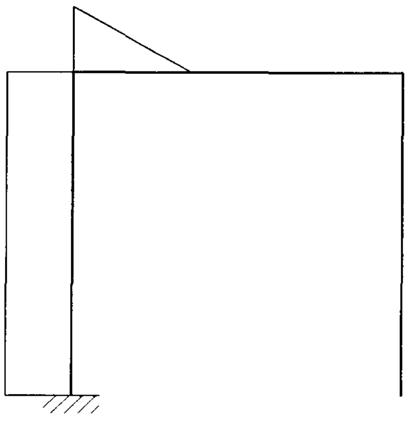


Figure 3 *Bending moment diagram for released structure*

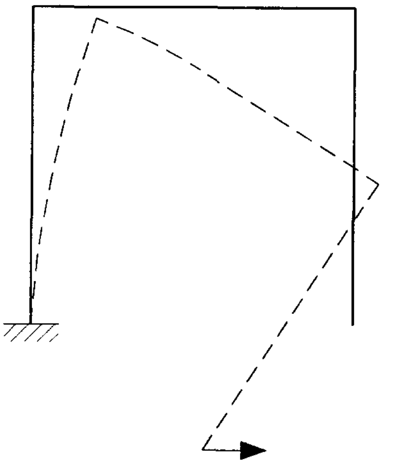


Figure 4 *Deflected shape of released structure*

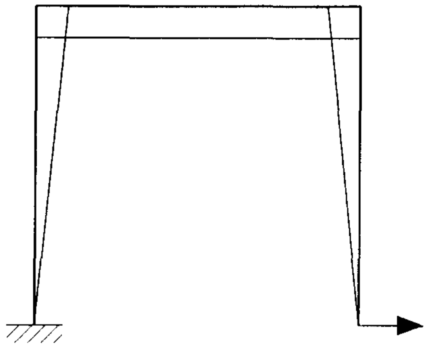


Figure 5 *Bending moment diagram for the reaction*

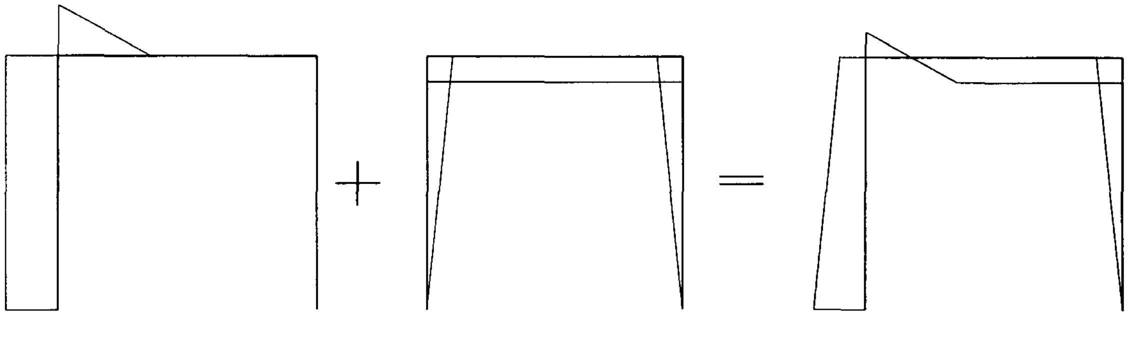


Figure 6 *Construction of the final bending moment diagram*

3.2 The influence of structural form on fabrication

In addition to an understanding of structural response, the structural designer must have an appreciation of practical fabrication and erection of steel structures. The work of a fabricator is primarily concerned with making connections between members, and activities associated with connections can account for up to 60% of the value added to the plain material. Simple connections are therefore one key to economic fabrication.

The process of structural design involves three stages:

1) Synthesis (conceptualisation)
2) Analysis
3) Design (the selection and checking of member sizes)

At the synthesis stage, the structural designer decides on the geometry of the structure, and how it is to resist the loads which are applied to it. Many consequences flow from the decisions about how the structure is to resist the applied loads:

- what analysis is necessary
- what the form of the joints in the structure will be
- how easy it will be to make
- how easy it will be to erect.

In the past, analysis of structures was a laborious manual task and decisions were made by the structural designer to reduce the effort required in analysis. These decisions generally resulted in a structure which was easy to construct, as well as easy to analyse and design. For example:

- making all beams simply supported avoids the need to distribute moments to other elements
- extensive rationalisation reduces the design effort necessary.

With the advent of quick and easy analysis by computer, the structural designer does not need to make simplifications to reduce analysis effort, and this can lead to more complicated, expensive, real structures.

During the synthesis stage of design, due regard should be given to the consequences of initial decisions on the fabrication and erection stages of the project. As an example, the economies in fabrication and erection of a pin-jointed braced frame, compared with a moment resisting frame, may mean that the simpler structure is chosen at the synthesis stage.

Guidelines on fabrication processes and relative costs are given in the CIMsteel publication *Design for Manufacture Guidelines*[2].

The benefits of an understanding of fabrication techniques is illustrated by the truss shown in Figure 7. When modelling the truss for analysis, the internal members may be rigidly connected at the nodes (the usual 'default' condition within most analysis programs) or released and modelled as pinned members.

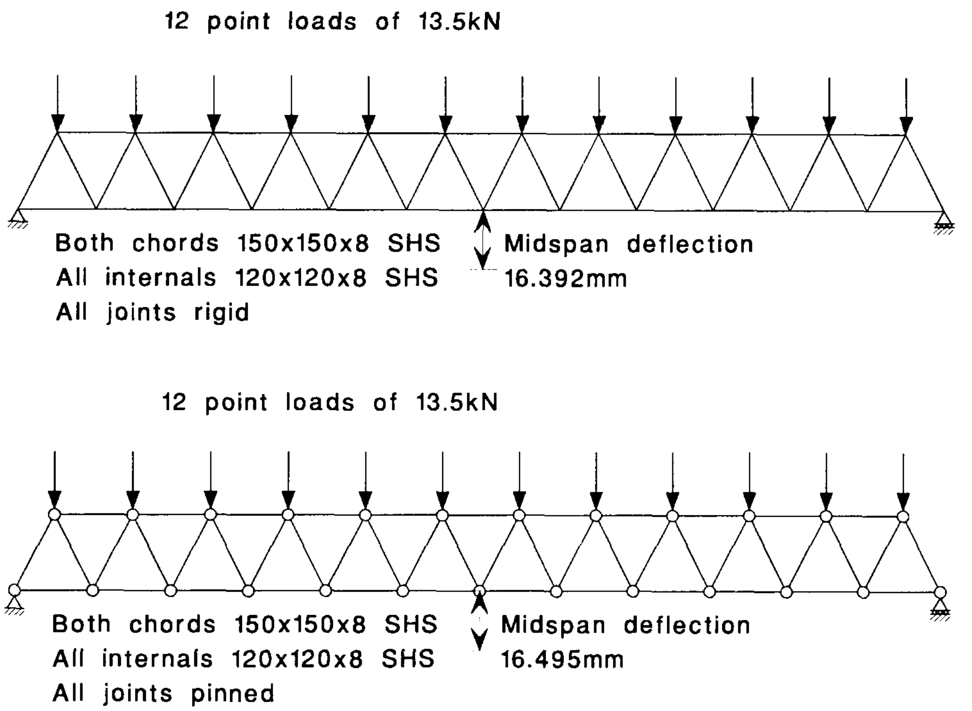


Figure 7 *Deflection of rigid and pinned models of the same truss*

Despite the fact that the structure is triangulated and the point loading is applied to the nodes, in the rigid frame there will be bending moments developed in the members because of the continuity at the nodes.

This simple example illustrates the following issues:

- To what extent should the analysis model be modified in order to simplify the fabrication?
- How does the structural designer recognise the implication of that change on the analysis model?
- How does the structural designer know what changes are appropriate?

In this case the issue is whether pinning the ends of the diagonals will significantly affect the behaviour of the truss. One way to make such a judgement is to compare the displacements. In the example analysed the central deflection differs only in the third significant figure between the pinned and rigid models; this is because the strain energy in the bending moments in the internal members is small. However, there is a significant difference in fabricating a moment connection at each node. As the pinned end model is a simple and cost-effective solution for fabrication, it should be the model adopted in the analysis and design.

3.3 Recommendations

Many examples could be quoted to illustrate where a more appropriate analysis model, or an appreciation of practical fabrication, would improve the cost effectiveness of a proposed design, if this were considered at the analysis stage. The structural designer must assess if an overall improvement in cost effectiveness could be achieved with a simpler model, or rationalisation of section sizes. The influence of connection details on the whole cost should never be forgotten. A least weight solution is rarely the cheapest overall, as illustrated by the form of the relationship between cost and weight in Figure 8.

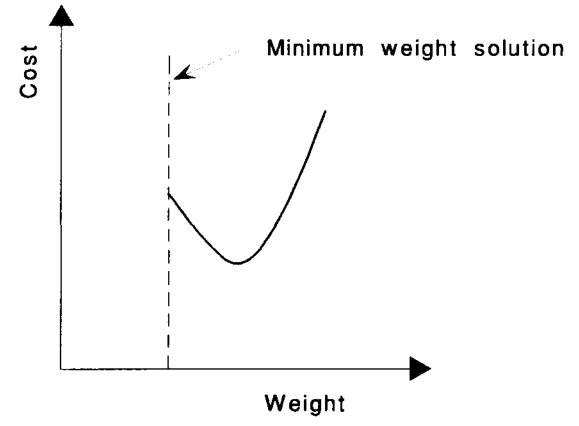


Figure 8 *Relationship between cost and weight of steel in typical frames*

The main objective is for the structural designer to improve 'buildability'. Most experienced structural designers do not have any difficulty in reaching a solution which achieves this goal. Those less experienced structural designers, working with reduced design time (due to fast track construction) and fee competition, and the facility to 'solve' by computer analysis the most complex analysis problem, have little opportunity to develop a qualitative approach or to understand the implications of design on fabrication and erection.

A qualitative approach can be developed by training and practice. The reader wishing to practice such skills is referred to Appendix A.

Advice on fabrication costs and cost effective details can be found in *Design for Manufacture Guidelines*[2].

Modelling issues and 'best practice' advice are found in the later Sections of this publication.

4 FRAME ANALYSIS

4.1 Types of analysis

There are three types of structural analysis programs in common use. These are:

- Elastic
- Plastic
- Elastic-plastic

Analysis programs do not include checks on the adequacy of members to sustain the forces generated. This is a 'design' activity and may be carried out by some software packages as a further process after the analysis.

Analysis may be first-order or second-order, as explained in the following sections.

4.1.1 First-order analysis

In first-order analysis the stiffness of the structure is assumed to be constant and unaffected by changes in the geometry of the structure when it is loaded. This is the standard assumption of linear-elastic analysis.

The principle of superposition applies to this approach. Where the analysis model remains the same, the results from analyses of different sets of applied loads can be added together and the results of individual load cases can be scaled. The analysis results are proportional to the applied loads.

4.1.2 Second-order analysis

In second-order problems, the effective stiffness of the structure is changed by the action of the loads upon it. Examples of this are cable structures, where a cable becomes stiffer as it straightens out, and a strut subject to axial load as well as lateral load. In the latter example the deflection under the lateral load is modified by the action of the axial load. The two load-effects interact and the principle of superposition does not apply. The flexural stiffness of an axially-loaded strut which is on the point of buckling is effectively zero.

Members acting in catenary as well as bending are similar to the cable example. In none of the examples given can the structural behaviour be modelled using a linear-elastic analysis.

Second-order effects are commonly illustrated by considering the additional displacements, forces and moments which arise from the action of applied loads on a deflecting structure. These are known as P-delta effects, and are demonstrated by example in Appendix B.

In some circumstances a first-order analysis may be used to approximate the results of a second-order analysis, by techniques such as the Amplified Sway Method, or by Extended Simple Design, which are both described in Clause 5.6.3. The Clause prescribes that in elastic design of multi-storey rigid jointed frames, either one of these approaches must be used when the frame is a sway frame, since the second order effects are significant. The Amplified Sway Method is suitable for elastic frame analysis by computer, and a brief worked example of this approach is included in Appendix C.

4.2 Elastic analysis

Elastic analysis programs are the most widely used for structural analysis and are based on the assumption that the material which is being modelled is linear-elastic. The appropriate value of the elastic modulus has to be provided in the analysis. The analysis does not take into account or check whether the elements can actually sustain the predicted load effects or whether they fail by yielding, buckling etc.

The majority of all software packages now allow the analysis of three dimensional structures although some may only allow two dimensional structures to be modelled.

4.3 Plastic analysis

Plastic analysis methods, more correctly called rigid-plastic, were commonly used in the analysis of plane-frame structures such as portal frames. The stress-strain curve for the material implicit in the analysis involves zero displacement up to the plastic resistance of the member, followed by continuous deformation at no increase in load (i.e. plastic collapse). Analysis programs based on this method are used to determine the set of plastic hinges which form a collapse mechanism for a given system of applied loads, and the load factor which corresponds to the mechanism. The mechanism which corresponds to the lowest load factor is used to design the frame.

These methods do not include the calculation of displacements and were usually applied only to two dimensional structures. For simple structures (for example single-bay portal frames), bending moments in parts of the frame other than those in which the plastic hinges have formed can be estimated from the plastic moments in the mechanism. Programs which carry out this form of analysis have now largely been superseded by those which perform elastic-plastic analysis.

4.4 Elastic-plastic analysis

First-order elastic-plastic analysis programs are often used for the analysis of portal frame structures. These programs assume that the elements behave elastically up to the formation of a plastic hinge and deform without sustaining further moment. The software is used to predict collapse mechanisms as in the rigid-plastic method of analysis but the analysis method allows the calculation of deflections and the value of bending moments in all the elements of the analysis model. This subject is dealt with in depth in Section 8.

Both portal frames and multi-storey rigid frames can be analysed using first-order elastic-plastic analysis methods to take into account the onset and formation of plastic hinges progressively around the frame, but second order effects (see Appendix B) must be properly considered. Second-order effects can be ignored if the frame is made sufficiently stiff such that the effects are negligible. Alternatively, second-order effects may be allowed for by using the Merchant-Rankine formula, or its modification by Wood. The latter is incorporated into Clause 5.7.3.3, and plastically designed multi-storey frames must comply with the provisions of this Clause.

Sufficient stiffness of plastically designed portal frames is ensured by the deflection and snap-through checks of Clause 5.5.3. Second-order effects in portal frames complying with the provisions of this Clause are small enough to be ignored.

A second-order elastic-plastic analysis would be required to take into account frame instability effects without recourse to the Merchant-Rankine method. Few analysis programs are capable of this.

4.5 Summary of analysis types

The types of analysis discussed in the preceding Sections are illustrated in Figure 9. The line representing linear elastic-plastic behaviour follows the linear elastic line until the formation of the first plastic hinge. This occurs at point A, where the bending moment at some point in the frame equals the plastic moment capacity of the section.

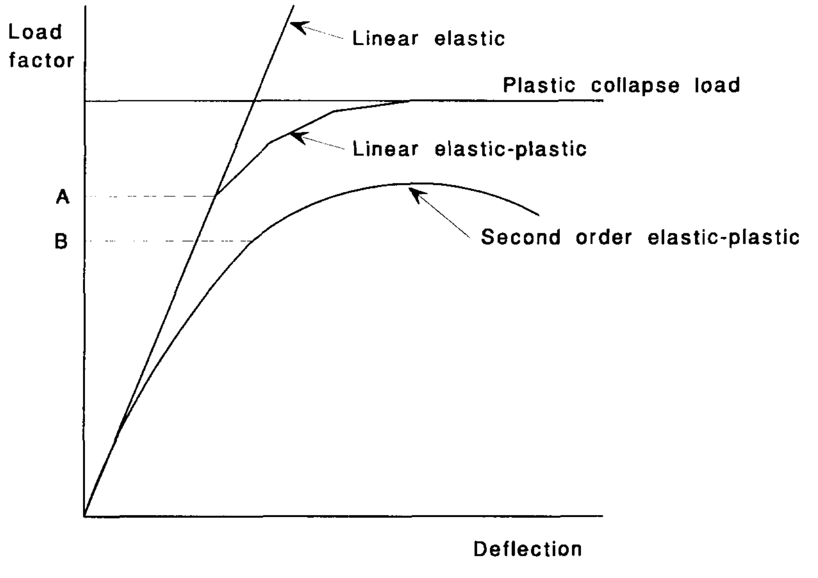
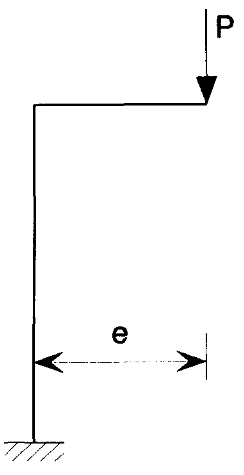


Figure 9 *Illustration of analysis types*

Linear elastic analysis may be continued past this level of load, although the resulting stresses in the members will exceed yield.

The straight sections on the linear elastic-plastic curve beyond point A represent the deflection of the structure between the formation of successive plastic hinges.

The second-order elastic-plastic analysis similarly follows the line of the second-order elastic analysis until the formation of the first hinge (Point B in Figure 9). The curve of the second-order analysis is explained by an examination of Figure 10.



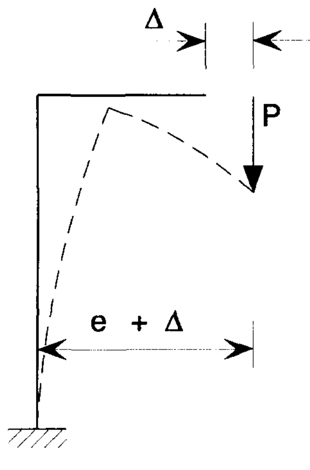


Figure 10 *Basic frame*

Figure 11 *Deflected form*

Point load P produces a bending moment Pe. According to a linear elastic analysis, P can be increased until (for plastic and compact sections) a plastic hinge is formed, such that:

$$Pe = M_p$$

The deflected shape of the structure is shown in Figure 11. The lever arm of the point load is increased, and hence for any given load, the bending moment predicted by a second-order analysis is greater than a first-order (linear) analysis.

Both these relationships are shown in Figure 12. This illustrates that in a second-order analysis, a hinge is predicted to form at a lower load level than in a linear analysis, which is also concluded from an inspection of Figure 9.

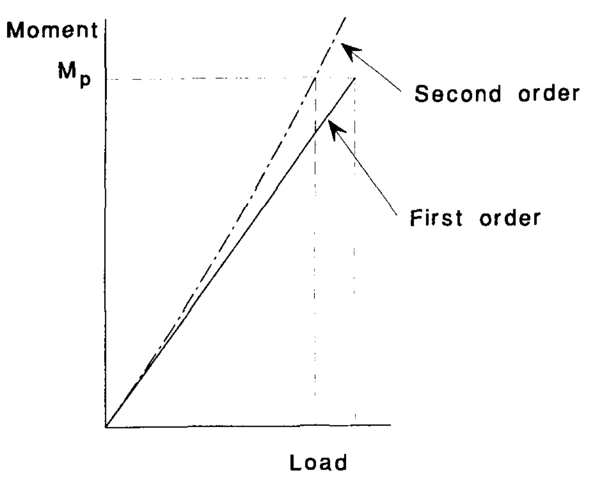


Figure 12 *Comparison of first and second-order analysis for the frame in Figure 10*

4.6 Inclusion of second-order effects in analysis

Second-order effects are particularly significant in certain structures, and must be included in their analysis.

Multi-storey frames in continuous construction may need to have second-order effects included, if the frame is a 'sway' frame, as described in Clause 5.1.3. For elastically designed multi-storey rigid frames which have been classified as a sway frame, a convenient method to allow for second-order effects is the Amplified Sway method, described in Appendix C. For plastically designed multi-storey frames classified as a sway frame, a simple approach is described in Clause 5.7.3.3.

Multi-storey frames in simple construction need not be checked for classification as 'sway' or 'non-sway', according to BS 5950: Part 1. However, the Foreword to BS 5950: Part 1 notes that the recommendations *apply to the majority of structures*, and assumes that bracing in simple construction will be well proportioned and robust. If the structure is of slender proportions, with an unorthodox bracing arrangement, second-order effects may be significant. Second-order effects will be significant if the frame is a 'sway' frame as described in Clause 5.1.3. A tall, slender building with unorthodox bracing (e.g. particularly steep bracing, or 'K' bracing) is an example of a braced structure which should be checked for classification as a 'sway' or 'non-sway' frame. If the frame is classified as a 'sway' frame, the Amplified Sway method illustrated in Appendix C can be used to amplify the lateral loading. In a braced frame, this will result in increased forces in the bracing members and the columns forming part of the bracing system.

The plastic analysis of portal frames in accordance with Clause 5.5 need not include second-order effects. Frames which satisfy this Clause are deemed to have geometry, stiffness and loading such that second-order effects are small enough to be ignored.

4.7 Analysis problems

When elements of widely different orders of stiffness are joined, the stiffness matrix is said to be 'ill-conditioned', leading to a loss of accuracy, despite the computational capability of the analysis software. If a mechanism is modelled, the stiffness matrix is 'singular', and usually cannot be solved. Analysis problems giving unexpected results are generally linked to one of these problems. It is impossible to list and describe all the problems here, but by far the most common is a lack of restraint in one or more directions or rotations, allowing the structure to form a mechanism or to 'spin'. In plane-frame analysis, this fault is uncommon, since most programs will detect the error, and most structural designers will provide restraints in this simple case. In three-dimensional modelling, the situation is altogether more complex, with what can appear a bewildering array of releases of members and supports.

Finally, reasonable results from a linear elastic analysis will only be obtained if reasonable stiffnesses are used in the analysis. Frequently, users of analysis programs tend to forget that the ratio of stiffnesses is critical. In the example of Figure 13, if the stiffness of the beam is many times higher than that of the columns, excessive deflections will result and the forces will not be correct. An exercise in The Structural Engineer(7) has shown that it is possible to model a multi-storey structure in which some parts of the frame deflect in the opposite direction to a horizontal load applied to one floor. This illustrates the care required when defining element properties in the analysis model. As a general guide, the ratio of stiffness of elements meeting at a node should not exceed 10^5.

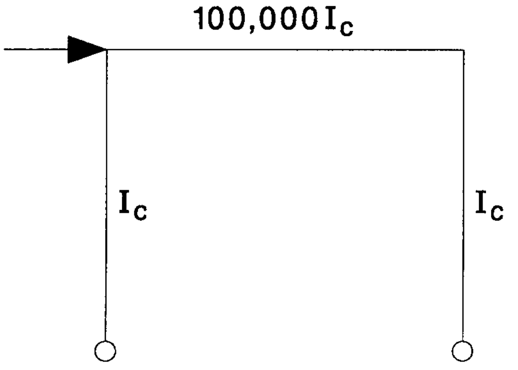


Figure 13 *Poorly conditioned structure*

4.8 Recommendations

First-order analysis is suitable for most orthodox steelwork structures, and is recommended for general use. Second-order analysis is generally not necessary for the analysis of straightforward, orthodox steel structures. Methods are available by which a first order analysis can be used to take into account second-order effects, where this is necessary. One such method, applicable to elastic design, is described in Appendix C.

Provision is made in BS 5950: Part 1 to ensure that structures are sufficiently stiff such that secondary effects are small enough to be ignored, or to ensure that the effects are taken into account. These provisions are described in the preceding Section.

Elastic analysis is widely and successfully used for most steel structures. Elastic-plastic analysis of portal frames is frequently undertaken by analysis software specifically written for this type of structure, and is used with equal success in practice. The modelling and analysis of portal frames is discussed in Section 8.

5 MODELLING OF FRAMED STRUCTURES

5.1 Introduction

It is generally convenient to consider first the form of the building frame in both sectional directions, and to identify:

- The **primary** structural elements which form the main frames and transfer both horizontal and vertical load to the foundations.
- The **secondary** structural elements, such as secondary beams or purlins, which transfer the loads to the **primary** structural elements.
- The **other** elements, such as cladding or partitions, which only transfer loads to the **primary** or **secondary** structural elements.

At the same time, any constraints on the form of the building must be identified as these may well dictate how the structure is modelled, and in particular, which (if any) frames may be braced, and which must be modelled as rigid.

The objective for the designer is (within the constraints of the specification and any architectural requirements) to provide a safe, economical structure. The definition of an economical structure is not straightforward, and it may be necessary to investigate several forms of framing before undertaking the detailed analysis and design. However, it is possible to provide general guidance based mainly on the understanding that moment resisting connections are significantly more expensive than nominal pinned connections. Thus in order of preference, the designer should consider:

- 'Simple' construction - i.e. braced frames with pinned connections
- Rigid frames in one direction, with ties and bracing in the other
- Rigid frames in two directions.

It must be emphasised that in most cases, there is more than one option for the form of the building frame. Further advice on structural form can be found in *Steel Designers' Manual*(8), and general guidance on economic details in *Design for Manufacture Guidelines*(2).

5.2 Two dimensional modelling

A number of advantages of two-dimensional models can be identified:

- simplicity of the analysis model;
- simplicity of the analysis output;
- a greater degree of rationalisation in member design and connection design.

These advantages have further practical benefit during fabrication, where economic benefit is gained with rationalisation, repetition and less complex connection details. For these reasons, a two dimensional frame with simple ties in the other plane is recommended.

Taking as an example the two storey office shown in Figure 14, it is clear that vertical bracing would not be allowed in every transverse frame. One solution is to model the transverse frames as rigid, but it may be possible to provide bracing on the longitudinal elevations. This would permit the structure to be modelled as a rigid frame in the transverse direction, with ties and bracing in the longitudinal elevation. This is a common form of such structures, and is of course similar to the normal form of portal frame buildings.

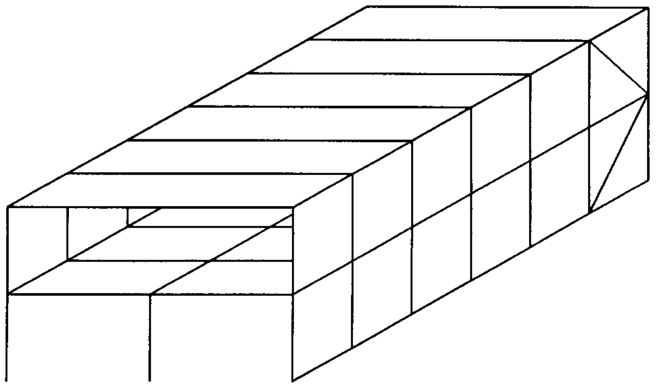


Figure 14 *Typical two storey office frame*

If, for some reason, bracing cannot be accommodated in the longitudinal elevations due to, for example, the presence of doors or glazing in each bay, discrete rigid frames are commonly provided in one or more bays to resist the longitudinal forces. The structure would still be modelled as rigid only in the transverse direction, with a separate model for the rigid frames on the elevations.

5.3 Three dimensional modelling

Three-dimensional modelling is undoubtedly a useful tool, particularly in complex structures which cannot easily be resolved into simple two dimensional frames. With a three dimensional model, it is also possible to transfer the complete design model into, for example, estimating software and detailing packages.

Three-dimensional modelling does, however, bring additional problems, with the potential to make mistakes within a large model, and with the complex analysis output which can frequently confuse, rather than elucidate. For orthodox structures, comprising two series of frames at approximately right angles, two dimensional modelling is usually satisfactory and is recommended.

The discussion in Section 3.2 on the influence of structural form on fabrication and erection is also relevant when considering the choice between three and two dimensional models. It is easy to define connections between beams and columns in orthogonal directions as rigid for analysis purposes, but the cost of the local stiffening required to achieve such connections in reality must be considered. In general, the cost of moment connections, particularly those between beams and the webs of columns, will outweigh savings made elsewhere in the structure. Three dimensional structures may be modelled with moment resisting connections in one orthogonal direction, and pinned connections in the other direction, if it is essential to model the entire structure.

5.4 Decomposition to two dimensional frames

'Decomposition' is the term used to describe the transformation of a real structure into the plane frame models used in analysis. The structural designer will generally produce both the structural form and the analysis model, based on intuition and experience, although general principles can be identified.

The first step is to identify the primary, secondary and other elements within the structure, as described in Section 5.1.

Having identified the primary, secondary and other structural elements, the second step is to eliminate the planar elements from the structure, such as cladding, facades and floor slabs. The next step is to eliminate the secondary elements, such as rails, purlins or secondary floor beams. At each stage, the applied loads are converted to equivalent loads acting on the reduced structure. Thus a roof load on a portal building will convert to a uniformly distributed load on the purlins as the roof cladding is eliminated from the model. As the purlins are eliminated, point loads will be applied to the rafter at purlin positions. In this example, following the recommendations of Section 12.2, the multiple point loads representing the purlins would be modelled as a uniformly distributed load on the rafter. The process of 'decomposition' is illustrated in Figure 15.

Certain obvious bracing has been omitted from Figure 15 for clarity.

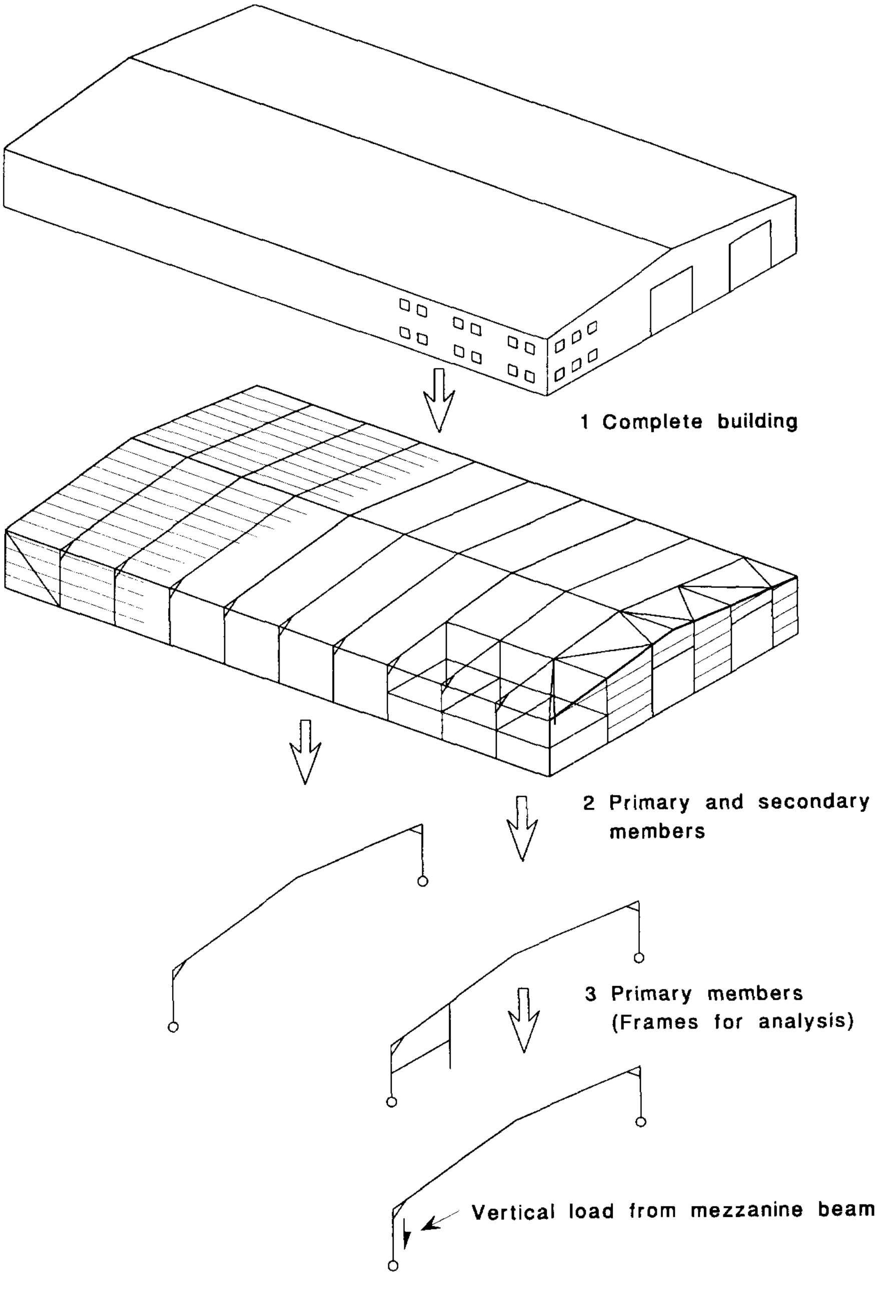


Figure 15 *'Decomposition' to main frames for analysis*

A careful consideration of the load paths is essential, to ensure loads from out-of-plane members are included in the analysis of the plane frame. Two common situations are shown in Figures 16 and 17. In the frame shown in Figure 16 the corner column carries axial load from the bracing system, and in Figure 17, the end reactions from the edge beams on the longitudinal elevations, and from the mezzanine floor, must be included.

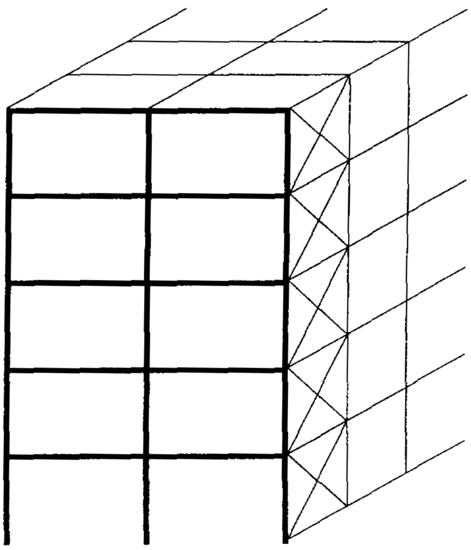


Figure 16 *Typical multi-storey frame*

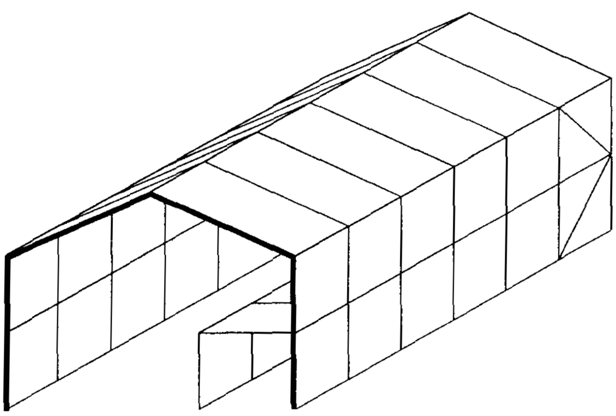


Figure 17 *Industrial building with internal first floor*

Most frames, being modelled as an intermediate frame within the length of a structure (Figure 18), will be subjected to load effects only in the plane of the frame. Note that the end frames (typically), although subjected to smaller in-plane loads, may also have out of plane loading to be taken into account. Out of plane effects (if any) can be included during the manual design of the members.

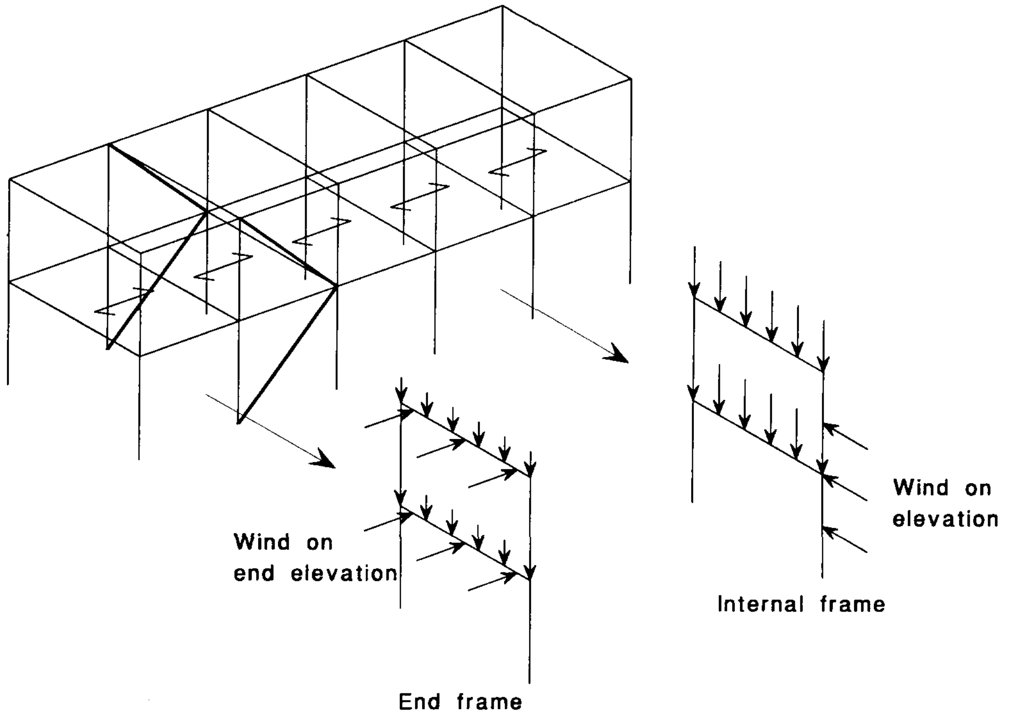


Figure 18 *Load effects on end and intermediate frames*

Although the secondary elements have been eliminated to produce a plane frame for analysis, the benefit of any restraint offered by the secondary elements may be utilised during member design.

In two dimensional modelling, it is generally assumed that out of plane members do not produce bi-axial bending or torsional effects in the primary frame. Thus the actual connections should be nominally pinned to minimise the out of plane effects which will, in reality, be present in all frames. In some circumstances, out of plane members produce bending effects in the primary frame which must either be included in the analysis, or included manually at the design stage. As an example, brickwork supports are usually subjected to torsion from the eccentric load, and the connections will be designed to transfer this moment into the primary structure. To model this, a bending moment (applied at a point) may be introduced in the frame loading. Alternatively, the additional moment may be included manually at the design stage.

5.5 Recommendations

Most common steelwork structures may be satisfactorily analysed as two dimensional models. Exceptions include some plant structures and structures designed to span in two directions such as space trusses.

Two-dimensional modelling is generally recommended, as:

- it is simpler than three-dimensional modelling.
- model frames are generally duplicated in reality, giving economy in analysis and design effort, and rationalisation and repetition in fabrication.
- connections to out-of-plane members are nominal pins wherever possible, avoiding complex and stiffened connections.
- most standard profiles are intended primarily for bending about one axis.

6 MODELLING OF BEAM AND COLUMN FRAMES

6.1 Basic frame geometry

It is normally sufficient to represent the frame by elements along the centrelines of the members, but questions can arise when discontinuities are present in reality. Two examples are shown in Figure 19. In the case of the column connection, a short, stiff horizontal member is introduced, which may be rigidly connected, pinned at either end, or centrally hinged. The choice of pin position affects the location and size of the resulting bending moment, and must be reflected in the connection design between the two sections. In many cases, option (i) will be appropriate, designing the lower column for an additional bending moment, and a nominally pinned connection from the upper column.

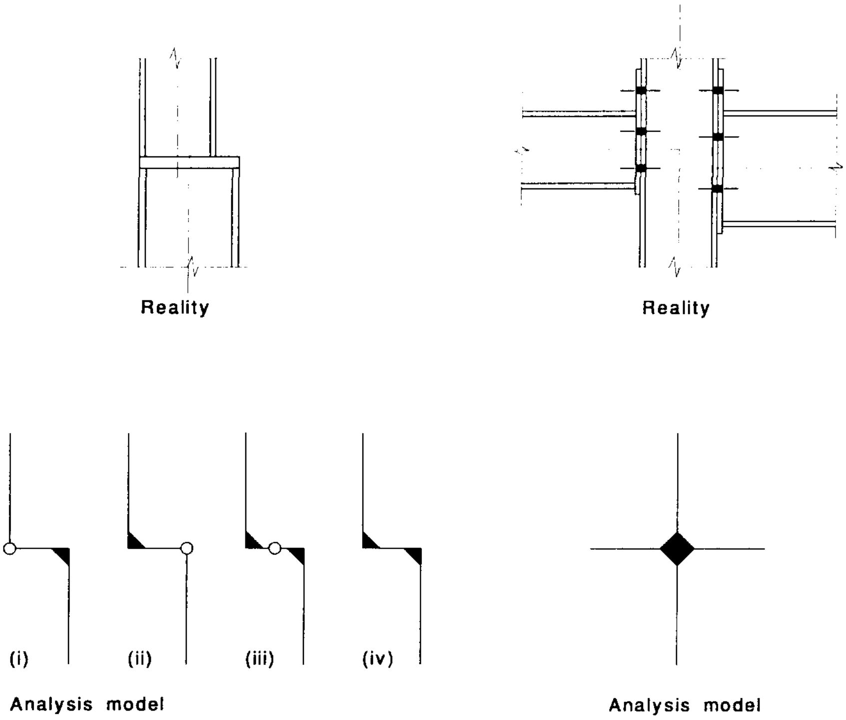


Figure 19 *Eccentric connections*

In the second example, it is not necessary to model the offset if the smaller beam remains within the depth of the larger, provided that any axial loads in the beams are small.

6.2 Composite frames

The reader's attention is drawn to the SCI publication *Commentary on BS 5950: Part 3: Section 3.1 'Composite Beams'*[9], where comprehensive advice may be found, and from which the following brief guidance has been extracted.

Currently, composite frame design involves a degree of manual redistribution of bending moments and manual design, compared to bare frame analysis and design which can be (and frequently is) carried out totally by software. Composite frame analysis is therefore usually carried out by considering a suitable sub-frame, such as shown in Figure 20, rather than a complete structure.

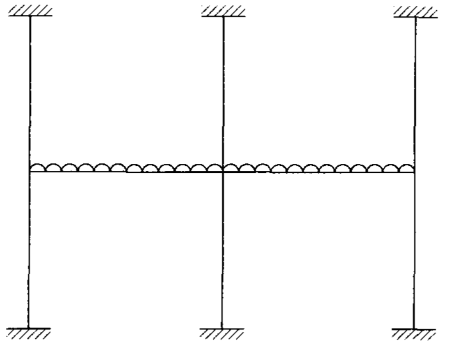


Figure 20 *Sub-frame for composite frame analysis*

Composite section properties must be 'transformed' into an equivalent steel section by dividing the cross-sectional area of concrete by the appropriate modular ratio, α_e. The relationship between composite section properties and steel section weight is shown in Figure 21, from which properties for initial analysis may be conveniently extracted.

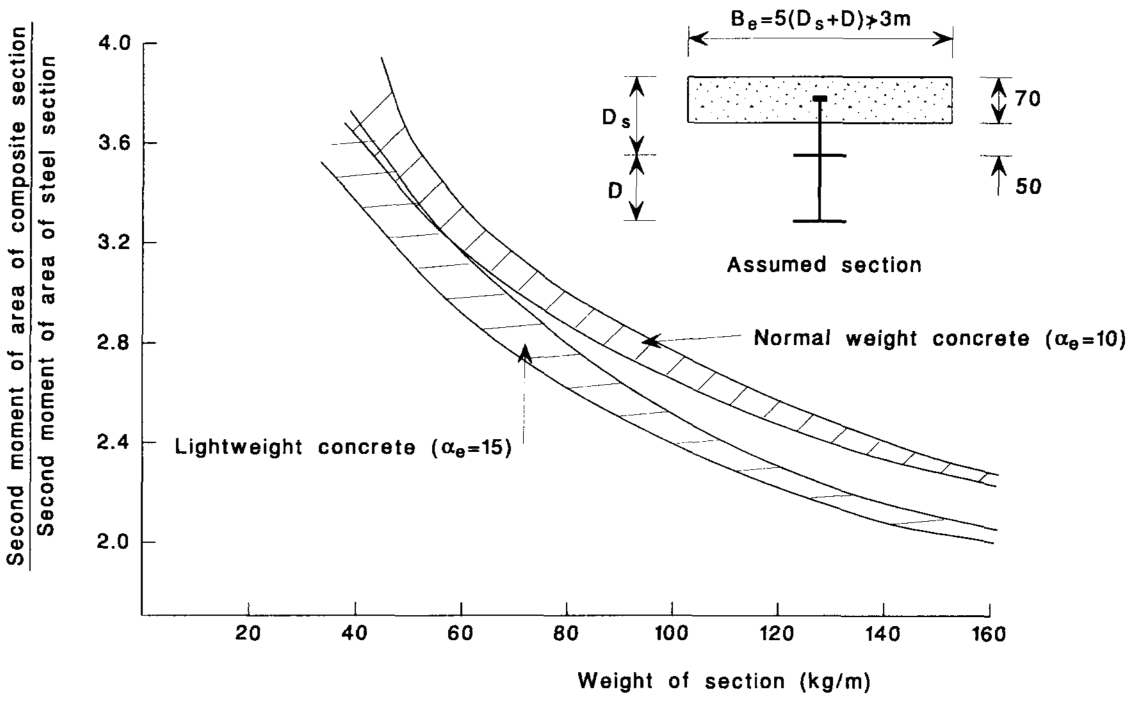


Figure 21 *Ratio of second moment of area of composite section to that of steel section*

An alternative approach, suitable for use only in computer programs, is to introduce for the initial analysis the composite section properties in the central 70% of the beam span. Bare steel properties (i.e. the cracked section) are used to the ends of each span, in the hogging moment regions. Following the determination of the bending moment distribution, the extent of the property types should be modified, and the frame re-analysed. This basic approach may be modified as appropriate when the bending moment distribution is asymmetric, for example in the case of sway frames.

In both cases, the moments may be redistributed depending on the method of analysis and the section classification at the support.

6.3 Shear deflection

Shear deflection occurs as a result of the shear strain in an element. It is an effect which is distinct from, and additional to, the bending deflection. The Engineer's theory of bending commonly used to calculate deflections by hand ignores shear deflection. Where non-composite beams are of the proportions normally used in the construction of conventional floors, neglecting the effect of shear in calculating beam deflection is acceptable because it is a small proportion of the bending deflection.

Where beams are deep in proportion to their length, shear deflection becomes more significant. Other situations where shear deflection may be a significant proportion of the total arise in composite construction where the bending stiffness of the beam may be very large because of the action of the concrete. The shear stiffness of the web of the beam is unaffected, and the defection due to shear therefore becomes a larger proportion of the overall deflection. The deflection of castellated and cellular beams also includes a relatively large component due to the action of shear.

If truss or Vierendeel frames are modelled by beam elements to simplify an analysis (or in preparing scheme calculations), shear deflection must be included in deflection calculations because in this case it is a significant proportion of the total. This is because the area of the members resisting the shearing force in a truss (the internal members) is a small proportion of the total area of the truss, whereas for a flanged beam the area of the web is significant when compared to the total area.

Shear deflection will be taken into account in an analysis if the beam element used in modelling the structure is formulated to do so. If an element is a 'shear beam', its 'shear area' will be required as an input to the program. This is because cross-sectional shapes vary significantly in their shear stiffness. Shear area is often input as a factor which is to be applied to the cross-sectional area of the element. The shear modulus, G, will also be required. For steel,

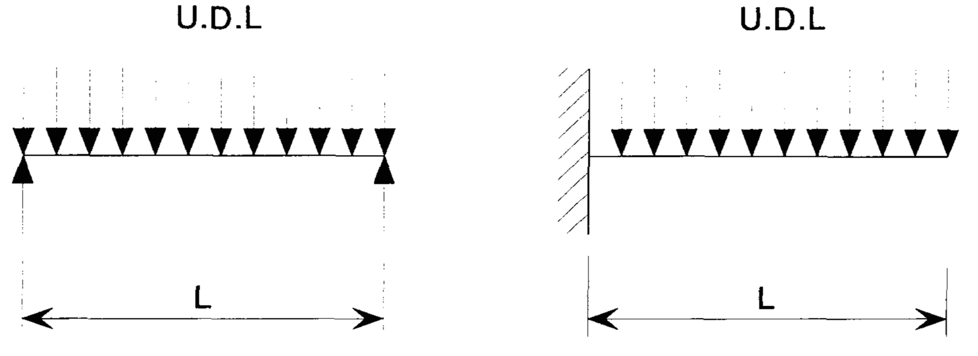
$$G = \frac{E}{2\,(1 + \nu)}$$

where: E = Elastic modulus
ν = Poisson's ratio (Taken as 0.30 in Clause 3.1.2.)

A requirement to provide this information indicates that shear deflection will be taken into account in the analysis.

6.3.1 Examples of shear deflection

As stated in Section 6.3, the effect of shear deflection increases as beams become 'stocky', i.e. they have a small span/depth ratio. In Figure 22 the additional deflection due to shear is shown as a percentage of the deflection due to bending, for simply supported and cantilever I section members. Minor approximations have been made to arrive at the figures shown in the tables.



Simply supported beam with U.D.L.	
Span/depth ratio	Shear deflection as % of bending deflection
20	3
10	14
5	54

Cantilever beam with U.D.L.	
Span/depth ratio	Shear deflection as % of bending deflection
20	1
10	6
5	23

Figure 22 *The influence of span/depth ratio on shear deflection*

Clearly the contribution of shear deflection to the total deflection becomes important only in the case of 'stocky' members. Note that plate girders have typical span/depth ratios of between 8 and 15. Span/depth ratios for universal sections in orthodox building construction are around 20.

In the examples illustrated, the contribution of shear deflection to the overall deflection is inversely proportional to the square of the span/depth ratio. At small span/depth ratios, shear deflection will exceed deflection due to bending, but both are likely to be small.

7 TRUSSES AND LATTICE GIRDERS

7.1 Analysis models

There are a variety of models which may be used for the analysis of a truss. These include:

- pin jointed frames.
- continuous chords and pin jointed internal (i.e. web) members.
- rigid frames.

The first two options are preferred since in most situations there will be no bending moments to be included in the joint capacity checks and connection design. In reality, secondary moments will be present, due to the change in geometry as the truss deflects, the actual rigidity at the connection and the stiffness of the members. Design standards generally define when these secondary stresses may be ignored. Clause 4.10 states that secondary effects may be assumed to be insignificant if the slenderness *(l/r)* of the chord members in the plane of the truss is greater than 50, and that of most of the web members is greater than 100.

An initial pin jointed analysis and preliminary member sizing is therefore required, to ascertain if the constraints on slenderness can be achieved. If, despite judicious choice of members, the secondary stresses cannot be ignored, a rigid frame analysis will be required to determine the bending moments in the members and at the connections.

7.2 Joint eccentricity

In most analysis and design situations, it is both convenient and reasonable that the connection design, being subsequent to both analysis and member design, is carried out in a manner consistent with the assumptions previously made. Thus in most situations, the type of connection (nominal pin, moment resisting etc.) follows the assumptions of the analysis.

In truss and lattice construction however, the design of the joints between chord and internal members frequently dominates the member design. Gap or overlap joints are often used to increase joint capacity, or to improve the fabrication detail. This introduces eccentricity into the setting out of the elements, and it will generally be necessary to include the effect of this eccentricity in the calculation of member forces and moments. As the eccentricities are not known prior to the choice of member, this is usually done manually, following an initial analysis with nodes at centreline intersections. If the eccentricities are known, they may be modelled as illustrated in Figure 23.

Certain types of connection have been well researched (notably those between hollow sections) and guidance exists that defines when the moment due to eccentricity at nodes may be ignored for connection design and design of some truss members. The guidance given by CIDECT[(10)] is reproduced in Table 1. Note that the design of the compression chord must always include the moments due to joint eccentricity. Further advice on joint capacities is to be found in EC3 Annex K[(11)] and also published by British Steel Tubes and Pipes[(12)].

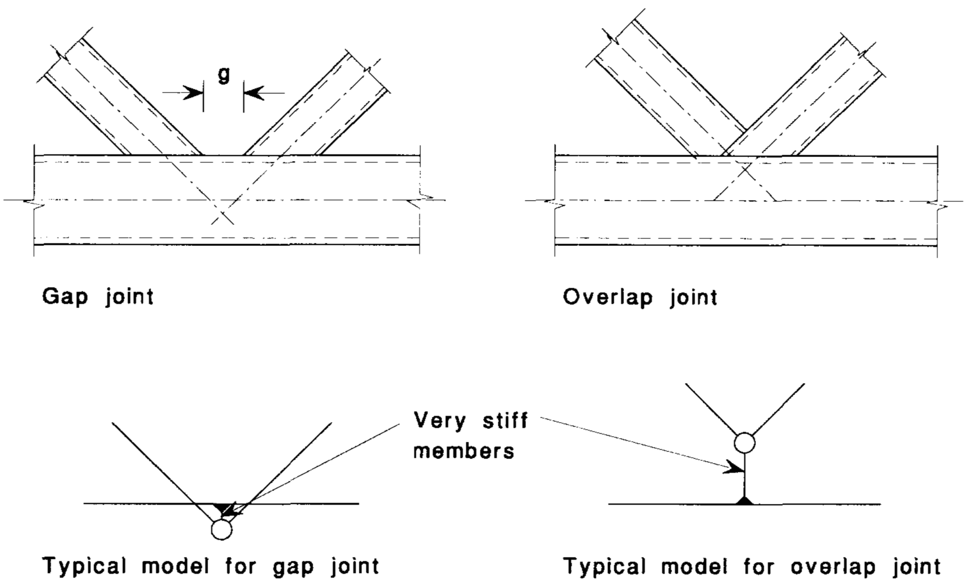


Figure 23 *Truss connections*

Table 1 *Guidance on when moments need to be considered for RHS truss design*

Type of Moment	**Primary**	**Primary**	**Secondary**
Moments due to:	Nodal eccentricity	Transverse member loading	Secondary effects
Compression chord design	Yes	Yes	No
Design of other members	No	Yes	No
Design of connections	No, provided eccentricity limits are not exceeded[10]	Yes[10]	No, provided validity limits are met[10]

7.3 Practical detailing

The modelling of truss and lattice structures is so dominated by member and connection design, that a few points of general advice are appropriate. In truss and lattice fabrication, connection details have a very significant influence on overall cost. Details which involve complex cutting, or extensive stiffening to improve joint capacities are to be avoided. The tempting idea of using an H section bottom chord (web horizontal) which will better resist out of plane buckling in a reversal load case, should only be adopted if the internal members can be satisfactorily connected. The difficult access for welding, size of gusset plate and load transfer through the web should be considered.

Similarly, changes in section size along the length of a chord are best avoided unless a cost effective, simple connection can be provided. If a complex, stiffened connection is necessary to transfer axial force and provide continuity of stiffness about both axes, it may be cheaper to maintain the same section throughout.

In hollow section truss construction, joint capacities are improved with chords that are smaller in size but with thicker walls and internal members which are relatively wide.

If the chords are I sections, with connections to the flanges, joint capacity is improved with thicker flanges, and internal members which are relatively narrow, compared with the width of the flange.

7.4 Truss analysis and design procedure

In summary, the analysis and design of a truss should be approached in the following sequence to obtain an efficient and economical structure.

i) Determine the truss layout, span, depth, panel lengths, lateral bracing by the usual methods, but keep the number of connections to a minimum, and maintain a minimum angle of 30° between chords and internal members.

ii) Determine loads; simplify these to equivalent loads at the nodes.

iii) Determine axial forces in all members by assuming that the joints are pinned and that all member centrelines intersect at nodes.

iv) Determine preliminary member sizes and check if secondary stresses can be ignored. If secondary stresses cannot be ignored, re-configure the truss or re-analyse the truss with rigid connections.

v) Check the joint geometry and joint capacities. Modify the joint geometry, with particular attention to the eccentricity limits. Consider the fabrication procedure when deciding on a joint layout.

vi) Check the effect of primary moments on the design of the chord members, using either the actual load positions or notional moments specified in the design standard. Add the effect of joint eccentricity where required, by manual methods, or by creating a new analysis model representing the actual setting out.

8 PORTAL FRAMES

This Section aims to define those areas of analysis modelling which are most often used in the computer aided analysis and design of portal frames.

Proprietary software dedicated to the analysis of portal frames generally involves an elastic analysis to check frame deflection at serviceability limit state, and an elastic-plastic analysis to determine the forces and moments in the frame at ultimate limit state. These methods have largely overtaken the rigid-plastic method.

8.1 Rigid-plastic and elastic-plastic analysis

8.1.1 The rigid-plastic method

The rigid plastic method is a simplified approach suitable for hand calculation and graphical methods, although it is also incorporated in some software.. In this method the frame is assumed not to deform under load (no linear elastic component) until all hinges required for a given mechanism have formed. The frame then collapses. The design process involves comparing a number of predetermined mechanisms to evaluate which one has the lowest load factor and hence represents the maximum load which could be carried by the frame prior to collapse. In each case, the bending moment diagram along the members is constructed to check that the plastic moment is nowhere exceeded.

For simple structures such as single span frames this process is a relatively simple matter since there are a very limited number of possible failure mechanisms.

However for more complex frames, e.g. multi-span, steps in eaves height, sprung supports or valley bases, the number of potential failure mechanisms, particularly under complex loading conditions, is vast. Alternative approaches are therefore usually incorporated to quickly establish a close approximation to the critical mechanism without the need to try all possibilities.

8.1.2 The elastic-plastic method

The elastic-plastic method, in addition to finding the collapse load, determines the order in which the hinges form, the load factor associated with each hinge formation, and how the bending moments around the frame vary between each hinge formation. The frame is assumed to behave linearly between each hinge formation.

The incremental approach of the method means that it can determine whether hinges form and later 'un-form' i.e. hinges cease to rotate and begin unloading as a result of the necessary redistribution of moment around the frame. This phenomenon and the incremental approach is best illustrated by an example. Consider the frame in Figure 24. The elastic-plastic analysis indicates that in this particular example, the first hinge would form at the sharp end of the haunch, B, at a load factor of 0.88. This can be confirmed by a linear elastic analysis since the frame remains elastic until the formation of the first hinge. The corresponding moment at the top of the stanchion, A, is less than M_p.

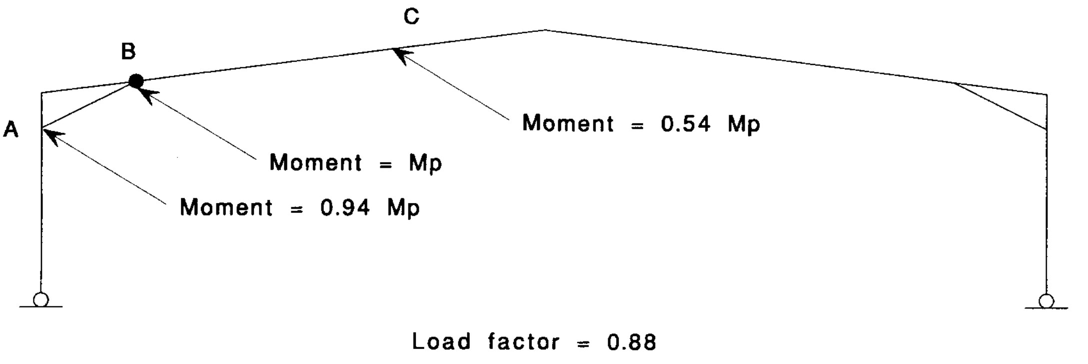


Figure 24 *Incremental approach - first step*

As more load is introduced, the next hinge to form is at the top of the stanchion at a load factor of 0.99 (Figure 25). Thus hinges now exist at positions A and B although as applied load is increased still further, the moment at hinge B would begin to reduce because of the continued redistribution of moment around the frame. This is known as hinge reversal.

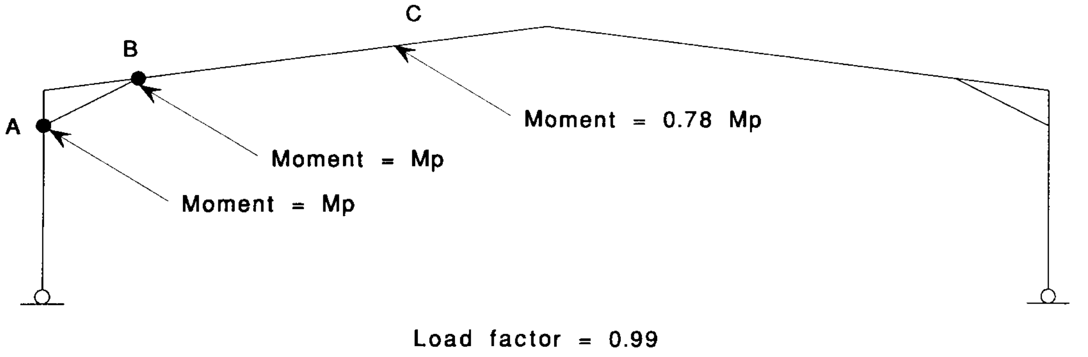


Figure 25 *Incremental approach - second step*

Finally the last hinge to form would be in the rafter close to the apex, C, at a collapse load factor of 1.05 (Figure 26). It may be noted that at Ultimate Limit State (load factor = 1.0), the moment at B will be very close to M_p and importantly will have undergone some rotation.

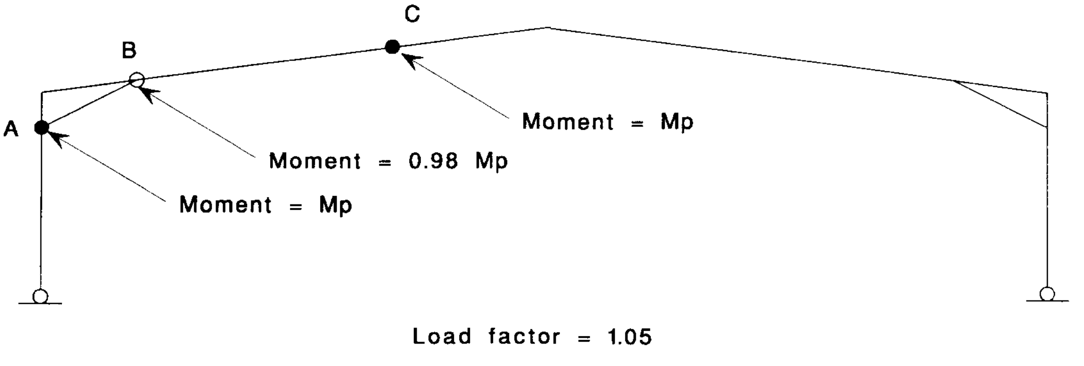


Figure 26 *Incremental approach - final bending moments*

Elastic-plastic analysis programs have largely replaced rigid-plastic ones for the following reasons:

- The state of the frame can be established at any load factor rather than only at collapse. This allows an accurate determination of the bending moment diagram at a load factor of 1.0, i.e. at ultimate limit state.
- Determination of the critical mechanism for more complex frames using the rigid-plastic method is not a simple matter and may lead to slight approximations. The elastic-plastic method will always find the critical collapse mechanism.
- The elastic-plastic method has a complete hinge formation history, whereas the rigid-plastic method takes account of only those hinges which exist at collapse. Therefore any hinge which forms, rotates, ceases to rotate and then unloads is not identified by the rigid-plastic method.

8.2 Rigorous implementation of the elastic-plastic method

It is perfectly possible to use a straightforward elastic analysis program in a 'step-wise' manner to produce a pseudo elastic-plastic analysis. This is relatively easy in conceptual terms but can be very tedious for anything but the simplest of frames. The process is an aid to understanding the way elastic-plastic analysis operates.

The first step is to carry out an elastic analysis at the full design loading. It is then necessary to investigate the bending moment diagram around the frame and determine the point or node at which the ratio of the applied moment to the plastic moment of resistance of the section is the greatest. This is the position of the first hinge formation. A new model is then created with a pin at that point, and a pair of equal and opposite moments equal to M_p of the section applied at the pin. This new model is then reanalysed to determine the position of the next hinge formation. A further pin and pair of moments are inserted at that position, the model reconstituted and the process continued.

This was the basic approach of early software for elastic-plastic analysis, although the re-creation of the model was incorporated internally within the software by reconstituting the stiffness matrix at each hinge formation. Computationally this was found to be inefficient and, as with the hand method, did not cope easily with complex features such as hinge reversal.

8.3 Haunches

Haunches are frequently provided at the eaves and apex connections of a portal frame. These should be modelled as tapered members, as recommended in Section 9.2.

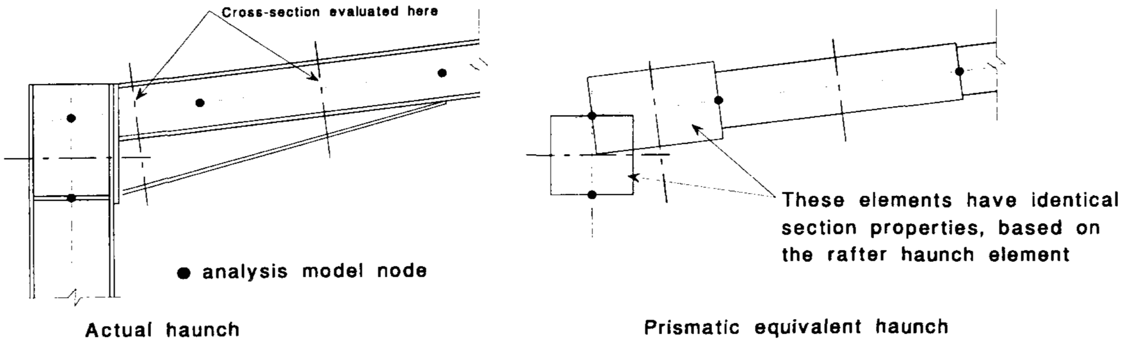


Figure 27 *Modelling of eaves haunch*

Eaves haunches of normal proportions may satisfactorily be modelled with two 'rafter' elements and one 'column' element, evaluated at the cross-sections shown in Figure 27. The haunched rafter is modelled with average section properties for lengths corresponding to ⅓ and ⅔ of the haunch as shown. The top of the column may be modelled using the section properties of the deeper haunch section. The assumption that the neutral axis remains at the centre line of the rafter and does not descend towards the haunch is safe, since it tends to overestimate both the compression in the bottom flange, and the shear.

Increased refinement is not justified by improved accuracy for most normally proportioned eaves haunches. The equivalent elements should be connected rigidly at their intersections.

Plastic hinges must not be allowed to form in the haunched region during the elastic-plastic analysis. Hence, when defining the properties for the haunch elements, the moment capacity should be set to a large value. (Depending on the software, this may be by direct input of a high moment capacity, or, for example, by input of a high section modulus.)

Apex haunches of normal proportions have no significant influence on the frame analysis, and do not need to be included in the analysis model. So-called 'apex' haunches in propped portals or monopitch portals make a significant contribution to the frame, and should be modelled in the analysis.

8.4 Portal bases

The modelling of bases is covered in detail in Section 11. The following points are particularly relevant to elastic-plastic portal frame analysis, where horizontal, vertical and rotational spring stiffnesses can be combined with a moment capacity.

- If portal bases are modelled with a high moment capacity and relatively low rotational spring stiffness, significant rotation would be necessary in order for a plastic hinge to form at the base.
- If portal bases are modelled with a low moment capacity and relatively high rotational spring stiffness, then it is likely that a hinge forming part of the final collapse mechanism will occur at a base position. Furthermore, it is likely that the moment at the base from the elastic analysis at Serviceability Limit State will be greater than the moment capacity of the base.

In reality, typical bases can only sustain an angle of rotation of less than 10°, and so it is important to choose a moment capacity and rotational spring stiffness which are reasonably balanced. If unbalanced cases occur, then the analysis results should be checked by hand or by the program to ensure that the base has not rotated (either elastically or plastically) by an unacceptable amount. Analysis programs may present a warning if a pre-set rotation limit is exceeded, or may indicate the calculated rotation at nodes for the user to check. It is particularly important in the case of low moment capacity and relatively high rotational spring stiffness to judge whether the plastic rotation which is inferred can be accommodated by the base detail, i.e. is the base sufficiently ductile? Embedded holding down bolts and the welds should not be relied upon to provide ductility.

Note that a moment capacity for a base has no relevance in an elastic analysis.

8.5 Valley supports

'Hit and miss' valley frames are common in portal frame buildings - this is where one or more internal stanchions in a multi-span frame are omitted in every second frame, the 'miss' frame. Valley beams running longitudinally support the rafters at the miss positions and react back onto the columns of the adjacent frames, the 'hit' frames.

In a typical two dimensional elastic-plastic analysis, valley supports are usually modelled by the inclusion of vertical, horizontal and rotational spring stiffnesses at those positions. Specifying a support moment capacity at a valley beam would imply plastic torsional behaviour of the valley beam, and this option is generally not available in proprietary portal frame analysis software.

The behaviour of the hit and miss frames are influenced by each other and so an iterative approach to the analysis and design of both is required;

- the reaction from the valley beam has to be included in the loading on the 'hit' frame but will be unknown until the 'miss' frame has been analysed and designed,
- the spring stiffness of the valley beam in the miss frame will be unknown until the beam has been designed or a section size estimated,
- the horizontal deflections of both frames need to be similar, since in reality the sheeting, which is very stiff, constrains the two frames to move together.

The vertical spring stiffness is relatively easy to calculate, knowing (or estimating) the section size of the valley beam. The horizontal deflection of the beam due to a unit point load can be calculated using Engineer's bending theory, and defined by the ratio of [deflection]/[force]. The spring stiffness is the inverse of this, in appropriate units. The horizontal deflection of the valley beam will depend on the degree of fixity assumed at the supports on the 'hit' frames.

A horizontal spring stiffness can be calculated in a similar manner using the weak axis properties of the valley beam. The horizontal supports to the valley beam (the 'hit' frames) are however not rigid, and the horizontal deflection calculation must include the support deflection before determining the equivalent spring stiffness. A horizontal spring stiffness will produce a horizontal load on the valley beam, requiring the valley beam to be designed for biaxial bending. Alternatively, bracing in the plane of the roof may be provided to the valley beam. In both cases the horizontal reactions must be included in the design of the 'hit' frame.

One convenient approach when using bracing to the valley beam, is to apply an assumed horizontal 'support' force at the valley of the 'miss' frame, which should be released horizontally. An equal and opposite force is applied at the valley of the 'hit' frame, and the horizontal deflections compared. This approach is then repeated, until the two horizontal deflections are approximately equal. The analysis of each frame including the calculated horizontal force will generate the correct forces, moments and deflections.

9 MEMBERS

Normal frame members are generally modelled as one (or more) straight elements, with associated section properties. Universal beams, universal columns, tees, angles, channels and hollow sections are modelled on this basis. The following sections give guidance on how to model non-standard sections.

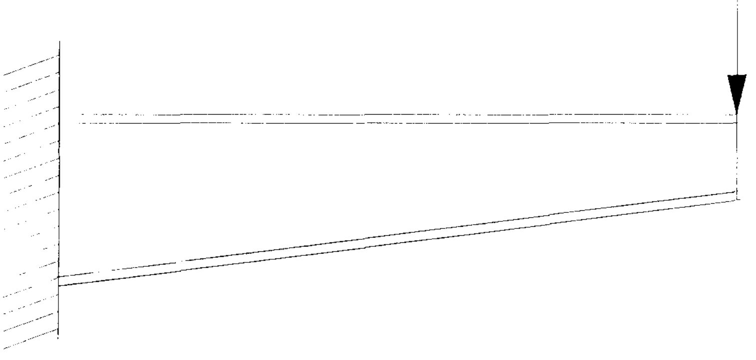
9.1 Curved members

Curved members are modelled as a series of short, straight elements. Modelling by using more, shorter elements, improves the accuracy of the results. As a general guide, a length of arc corresponding to 15° produces reasonable results.

9.2 Tapered members

Tapered members can be simply modelled as a series of short elements, each with an inertia corresponding to the depth of the member at that position. Two to five such sections give reasonable accuracy.

Figure 28 shows a simple cantilever, with an inertia I at the tip, and an inertia of $4.6I$ at the support. If the cantilever is modelled with three, five, and ten sections, the following results are typical. Taking the deflection of a model with five sections as the standard, using three sections modified the tip deflection by 2%, and ten sections by 1%, as shown in Figure 28. Note that modelling the cantilever as a single member with an average inertia of $2.8I$ gives a tip deflection some 15% different from the stepped member. Generally, three sections are satisfactory when modelling tapered members.



Number of elements	Deflection at tip (%)
1 (average inertia)	115
3	102
5	100
10	99

Figure 28 *Deflection of a cantilever with different element inertias*

Some programs include facilities for modelling tapered members, but they usually follow the approach described above.

9.3 Stiffness of stub elements

The stiffness of stub elements introduced in bracing systems, frames or trusses must be carefully chosen or the analysis may yield inaccurate results.

As a general guide the problem can be avoided if the stiffness (*I/L*) of elements meeting at a node do not differ by more than a factor of 10^5. This will be satisfied in most cases if properties of rolled sections are used as stubs in preference to creating an element with a massive inertia.

Some programs have the facility to provide a rigid link between members meeting with a small eccentricity, and it will not be necessary to determine a suitable inertia if this facility is available. Similarly, members meeting in this fashion may be 'coupled', which allows the release options described in Section 6.1.

9.4 Castellated and cellular members

Many steelwork analysis programs provide libraries of standard section properties, and may also include the section properties for castellated beams. This will allow the structural designer to include castellated members in a frame model in the same way as standard sections. Whilst the frame bending moments produced by this approach will generally be satisfactory, the structural designer should note that the deflection of a castellated or cellular beam will be more than that predicted by Engineer's bending theory. This is due to the Vierendeel effect and to shear deflection (see Section 6.3).

As a rule of thumb, the deflection of a cellular or castellated beam may be taken as 25% greater than the equivalent depth beam without openings. Additional deflection due to the Vierendeel effect becomes more significant with multiple, long openings. As a rule of thumb, the deflection of a beam with multiple, long openings may be taken as 35% greater than that of the equivalent depth beam without openings.

In some circumstances, the structural designer may conclude that the additional deflection may be ignored, or is not critical. Alternatively an allowance may be made for the additional deflection during design and checking of the members.

Castellated and cellular beams may themselves be modelled as a frame, and detailed guidance will be found in EC3 Annex N[(11)].

10 MODELLING OF CONNECTIONS

10.1 Connection behaviour

Within a frame, connection behaviour affects the distribution of internal forces and moments and the overall deformation of the structure. In many cases, however, the effect of modelling a stiff connection as fully rigid, or a simple connection as perfectly pinned, compared to modelling the real behaviour, is sufficiently small to be neglected. Elastic analysis programs consider only the stiffness of the connection and it is convenient to define three connection types as follows:

Rigid
A connection which is stiff enough for the effect of its flexibility on the frame bending moment diagram to be neglected. In practice, the flexibility (rotational stiffness) of a connection is not usually determined. Connections designed on a strength basis alone are generally considered to be rigid.

Semi-Rigid
A connection which is too flexible to qualify as rigid, but is not a pin. The behaviour of this type of joint must be taken into account in the frame analysis.

Pinned
A connection which is sufficiently flexible to be regarded as a pin for analysis purposes.

Connections with a capacity of less than 25% of the beam moment capacity, together with some ductility or freedom to rotate, are commonly regarded as pinned. In practice, certain details, typified by those found in *Joints in Simple Construction, Volume 2*[13] are considered to be pinned, and no calculation of the connection stiffness is attempted.

Typical examples of 'rigid' and 'pinned' connections are shown in Figure 29, and moment-rotation curves are illustrated in Figure 30.

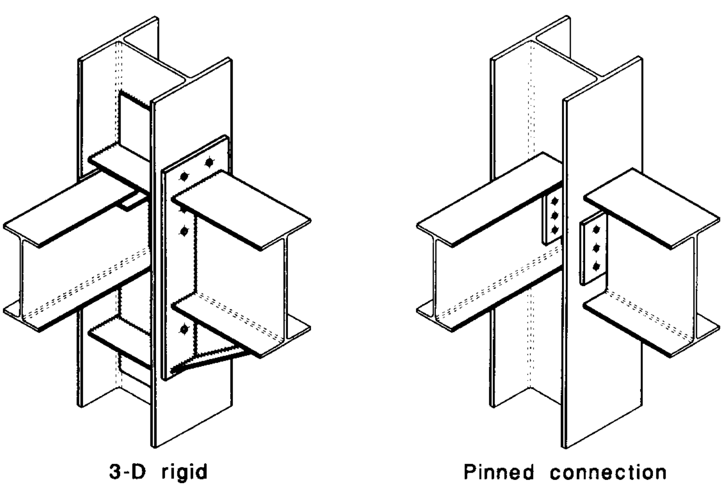


Figure 29 *Typical connections (UK practice)*

10.2 Rigorous approach to the modelling of connections

Whilst the common technique of modelling connections in analysis as absolutely rigid or totally pinned has been successfully applied for many years, a rigorous approach would acknowledge that all 'rigid' connections exhibit a degree of flexibility, and all 'pinned' connections possess some stiffness. To adopt a rigorous approach to joint modelling, two questions need to be resolved by the structural designer:

1. What are the limits which define a rigid, pinned or semi-rigid connection?
2. How stiff is the particular connection?

These two issues are considered in the following two sections.

10.3 Stiffness limits

Figure 30 shows a number of moment-rotation curves, representing connections of varying stiffness, and shows the dividing lines between rigid, semi-rigid and pinned connections.

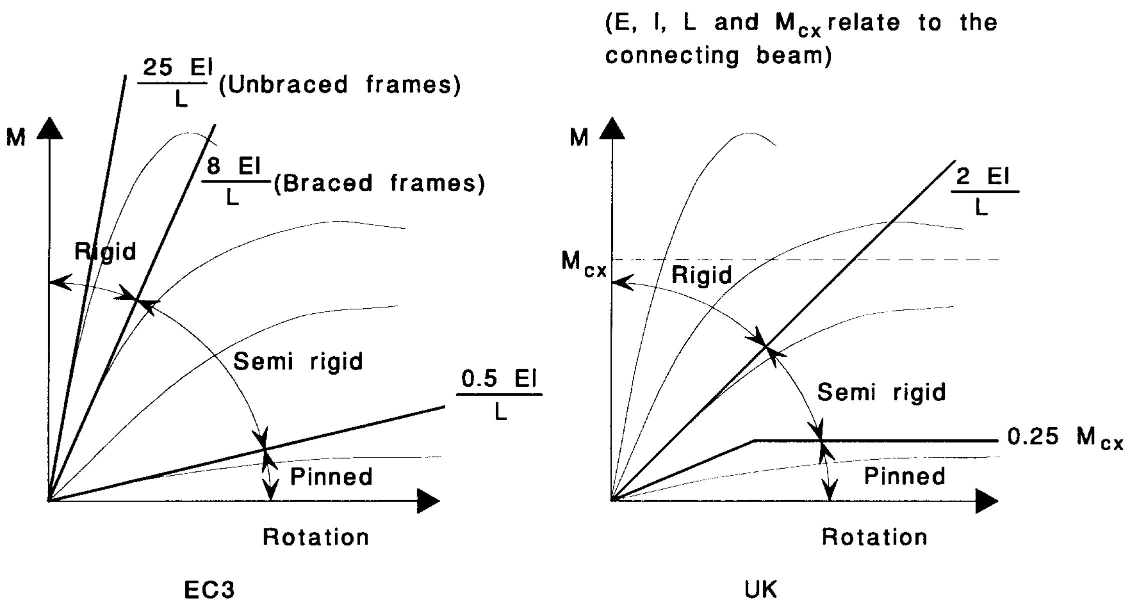


Figure 30 *Stiffness limits*

Unfortunately, there is no common agreement on the slope of these dividing lines. Within the UK, the figure of $2EI/L$ has been suggested as the division between rigid and semi-rigid. However EC3 Annex J[11] offers 2 alternatives:

$8EI/L$ for braced frames, and

$25EI/L$ for unbraced frames.

In Annex J, the slope of the line between pinned and semi-rigid connections for both braced and unbraced frames, is given as $0.5EI/L$. In the UK, however, pinned connections are usually defined by their moment capacity, not stiffness. Connections which have a maximum capacity of less than 25% M_p are generally regarded as pinned, provided they have some ductility or freedom to rotate.

10.4 Assessment of actual connection stiffness

The only accurate way at the present time to determine the moment-rotation characteristics of a connection is by testing. Methods of calculating connection stiffness do exist, notably in EC3 Annex J[11]. Many structural designers have little confidence in the predictions made in the current (1995) version of Annex J, when compared to test results. Assessments of connection stiffness are therefore usually subjective.

10.5 Modelling of semi-rigid connection behaviour

Due to the uncertainties described in Sections 10.3 and 10.4, it is relatively uncommon to determine connection stiffness prior to, or during analysis. In particular, frame analysis with springs representing connection stiffness is uncommon, although some specialist programs can include a stiffness function which varies with the applied moment. If the connection characteristics are known, or can be calculated, procedures do exist for incorporating the effects of connection flexibility into standard methods of frame analysis[14].

Annex J presents a general connection model shown in Figure 31, with flexural springs for each beam connection and a diagonal translational spring to represent the web panel. The difficulty still lies in determining the various stiffnesses required and having described the model, Annex J states that this type of modelling is not considered further. Modelling a connection in this way is not recommended.

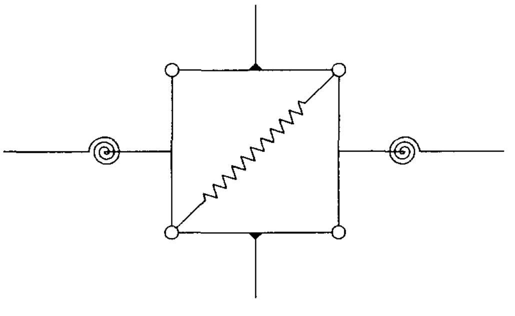


Figure 31 *Connection model (EC3 Annex J)*

A 'simplified' approach is presented in Annex J without the web panel spring, by modifying the beam connection stiffness. The Annex also gives a method for predicting the initial connection stiffness for certain connections. The concerns raised in Section 10.4 still remain, however.

For analysis purposes, Figure 32 illustrates that the stiffness varies with changing values of moment. Annex J overcomes this by allowing the initial connection stiffness to be used up to an applied moment of two thirds of the connection resistance, and modifying the stiffness if the applied moment is more than this limit.

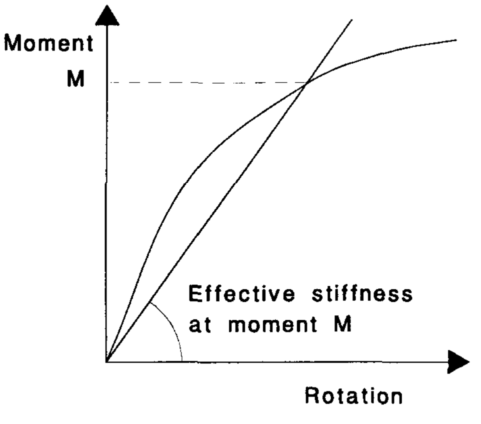


Figure 32 *Connection behaviour and effective stiffness*

10.6 Recommendations - connection behaviour

Modelling of semi-rigid connection behaviour is currently not recommended, although it is acknowledged that as predictive calculations become more reliable, and possibly more straightforward, modelling in this way may become more attractive. Advances in software, in particular the simple entry of connection characteristics, may make the modelling of semi-rigid connections a standard technique in the future, although the overall benefit of such an approach may need justification.

The usual practice of defining a connection in the model as rigid or pinned is recommended, with the connection design following the assumptions made in analysis.

The use of proven, simple connections (i.e. nominal moment capacity) as found in *Joints in Simple Construction*[13] is recommended for pinned connections.

For normal structures, well proportioned connections designed for strength alone may be assumed to be rigid. The exception to this is in multi-storey unbraced frames, where the connection rotational stiffness is inherent to the safety of this type of frame. The reader is referred to Clauses 5.6 and 5.7, and to *Joints in Steel Construction : Moment Connections*[15], where a degree of practical guidance is given to ensure the connections are rigid.

10.7 Modelling of connections

EC3 Annex H[11] states that connections should be modelled for global analysis in a way which appropriately reflects their expected behaviour under the relevant loading and suggests that connections may be modelled by:

(a) Nodes at the intersections of the member centrelines

(b) Nodes offset from the member centrelines to reflect the actual locations of the connections

(c) Special deformable connection elements of finite size

This would allow, for example, the support of a pin ended beam to be located

(a) At the centreline of the column

(b) At the face of the column

(c) At the centroid of a group of bolts or welds connecting the beam web to supporting cleats or brackets

(d) At the centreline of a supporting bracket under the beam.

Modelling in accordance with (a) above is common, as frequently the size of members or connection details will not be known at the first stage of analysis.

Connection design must then be consistent with the assumptions made in the analysis. In the common case of beams with pinned connections to columns, Clause 4.7.7 requires nominal moments to be applied to columns, calculated from eccentricities defined in Clause 4.7.6. Some design programs include this facility in the design module. Alternatively, if the designer is prepared to make a judgment on the probable column size, the nodes may be situated eccentric to the columns, thus producing the final moments in the column lengths. The short stubs from the column centre line are generally modelled as the beam section (Figure 33).

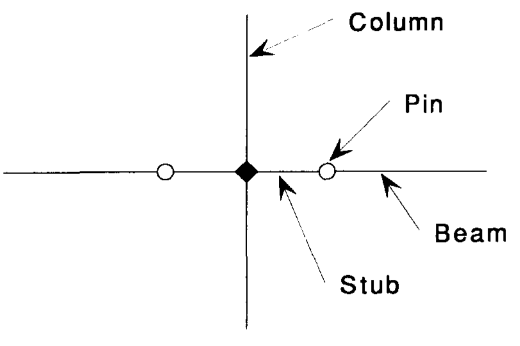


Figure 33 *Model of eccentric connections to column*

The modelling of connections some distance from the column centreline is always important when the actual details involve rigid stub members from the column member, and in these cases, the analysis should reflect the real detail.

A typical example of this is the hollow section beam to column connection shown in Figure 34.

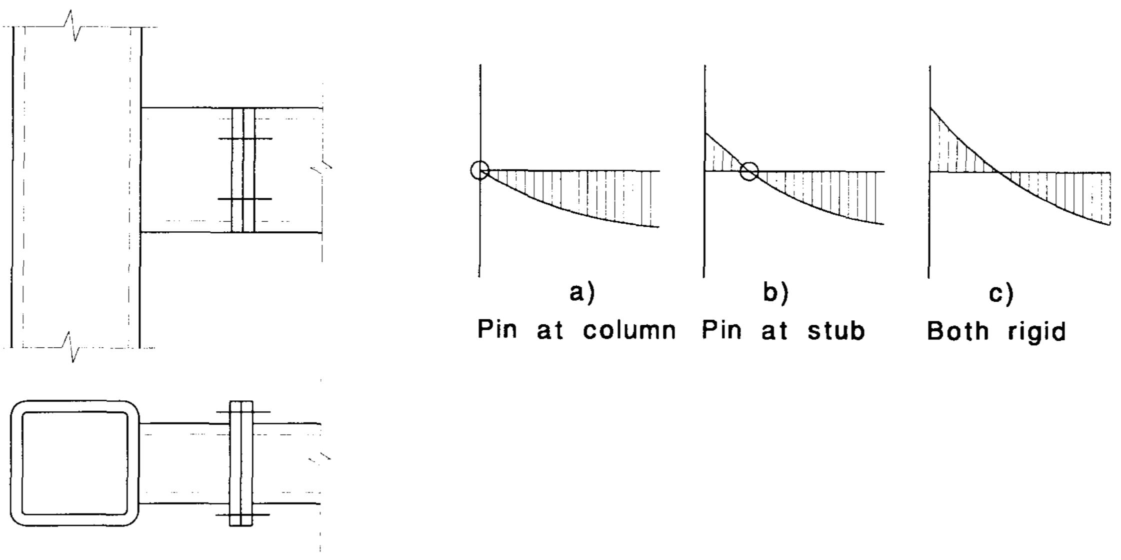


Figure 34 *Connection between hollow sections, with alternative bending moment diagrams*

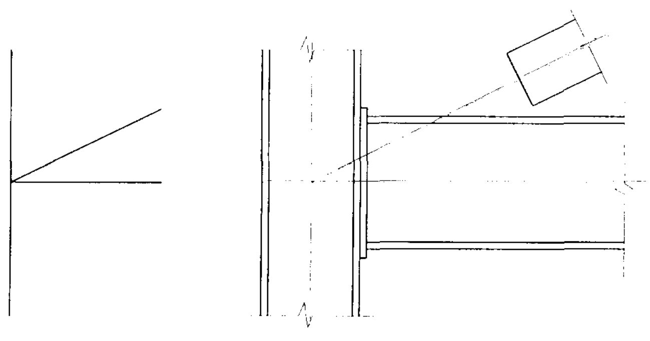
For economy and ease of erection, the beams shown in this example are not to be site welded to the columns, but bolted via end plates. If similar beams are present on all four faces, the connection must be made some way from the face of the column. The two alternative bending moment diagrams are indicated, with the following features, either:

- The flange plate beam to beam connection could be rigid, with a pinned connection at the column face, (bending moment diagram (a)), or
- The flange plate beam to beam connection could be a pin, with a rigid connection to the column, (bending moment diagram (b)), or
- Both the flange plate and beam to column connections could be rigid (bending moment diagram (c)).

In practical terms, one may perceive the welded connection to the columns to be more 'rigid' than 'pinned' and the second option (b) the correct model for analysis. The beam to beam conn would then be designed for shear alone, and detailed as appropriate for a pinned connection. An alternative (but equally valid) assessment, particularly with a relatively thin column wall, may be that the local bending stiffness at the face of the column constitutes the most flexible part of the connection.

10.8 Bracing connections

The modelling of bracing systems is frequently not straightforward and often leads to misunderstanding between the structural designer carrying out the analysis and the connection designer. The bracing, columns and floor beams are generally modelled on centreline intersections, as shown in Figure 35.



a) Model b) Reality

Figure 35 *Bracing arrangement*

The first point of confusion concerns the end reactions of the floor beam which in the output from the analysis will only contain (assuming a pinned connection) the shear forces from the applied floor loads, together with the axial force from the bracing system. In reality, the vertical shear in the beam-to-column connection should include the vertical component of the bracing force. The horizontal component of the bracing force is transferred directly to the beam, and not via the end connection.

Resolving the bracing force into horizontal and vertical components at the connection to the beam, further illustrates that both components of force induce bending in the beam which is not present in the analysis based on centreline intersections. The more serious effect is probably that the output for the beam end reaction is likely to omit the force components from the inclined bracing. With particularly shallow or steep bracing angles, more appropriate models for analysis are shown in Figure 36 (a) and (b). Stocky members with high inertia should be used for the stubs from the main members to the bracing. The bracing connections to the stubs should be modelled as pins, with rigid connections between the stubs and the main frame elements. A further alternative is to set out the bracing to the face of the column, (Figure 36(c)), which can often simplify the connection, but adds a bending moment to the column.

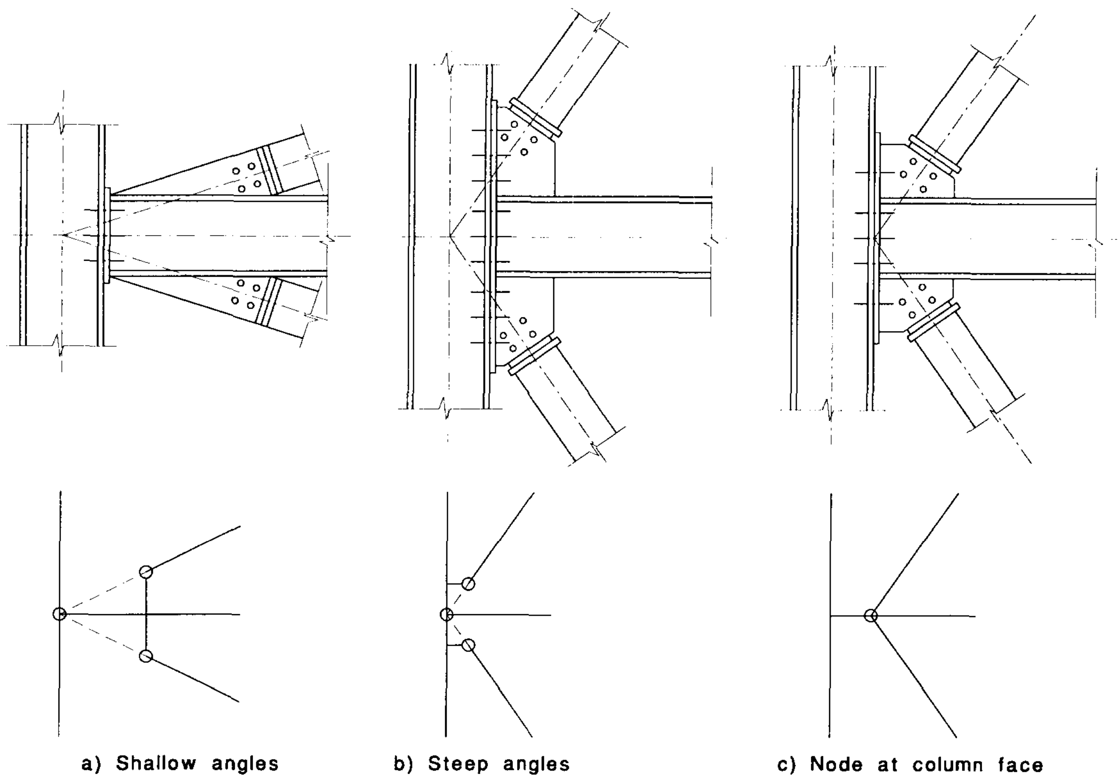


Figure 36 *Models for bracing*

10.9 Recommendations - modelling of connections

In general, nodes at centre line intersections are recommended in the analysis model for beam and column structures. Real eccentricities, if present, should be taken into account during member design.

Since member sizes are generally unknown at analysis stage, bracing should still be set out with nodes at the intersections of member centrelines for the initial analysis. A second analysis may be completed with stub members between bracing and main members, or alternatively the real effects may be included manually. The latter approach is recommended. The structural designer must ensure that the design loads for the connections are clearly conveyed to the connection designer. This may involve quoting loads in appropriate combinations, since, for example, the load factors applicable to the floor loads carried by a beam will reduce when wind load is included in that particular ultimate loadcase. Unrealistic combinations of connection forces lead to expensive connections.

11 MODELLING OF SUPPORTS

11.1 Rotational fixity

The interaction between the foundation and supporting ground is complex. Detailed advice is presented in the Institution of Structural Engineers publication *Soil-structure Interaction - The Behaviour of Structures*[16]. This report also contains examples of the complex way in which this interaction can be modelled, which is very probably too involved for general analysis. BS 5950: Part 1 has recommendations covering rotational stiffness which are suited to most situations.

Clause 5.1.2.4 states that in the absence of detailed knowledge of the foundation stiffness, the following should be assumed:

(a) Where the column is rigidly connected to a suitable foundation, the stiffness of the base should be taken as equal to the stiffness of the column, except as in 5.7.3.1. *(Note that the reference to Clause 5.7.3.1 is to preclude the calculation of effective column lengths smaller than 1.0L, not to override the need to allow for foundation stiffness when determining* λ_{cr}*).*

(b) Where the column is nominally connected to the foundation, a base stiffness of 10% of the column stiffness may be assumed.

(c) Where an actual pin or rocker is provided, the base stiffness should be taken as zero.

Despite the apparent clarity of (a) above, it is important to realise that the base stiffness has to be treated as a beam stiffness, not a column stiffness. This is more fully explained in *Steel Construction Today, November 1991*[17].

In many cases it can be visualised and modelled for analysis by rigidly connecting a beam member to the base as shown in Figure 37. If the dummy beam is given a length and inertia identical to the column, with the beam end remote from the column fixed, this will achieve the required base stiffness.

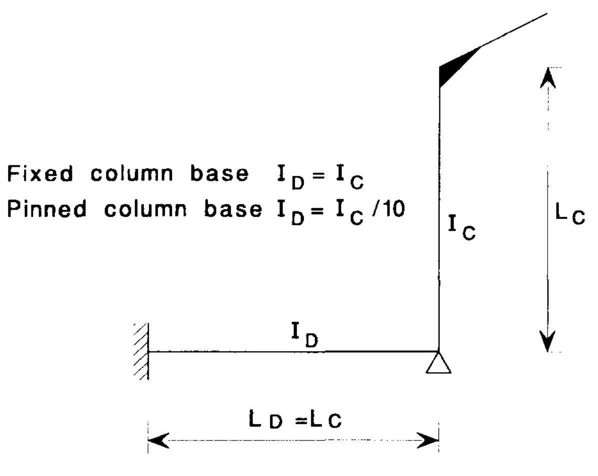


Figure 37 *Modelling of base stiffness*

In order to reduce the confusion caused by the moment at the end of the dummy member, it is more convenient to pin the remote end of the dummy beam as shown in Figure 38, and reduce the length to 0.75 × column length.

In both models, the inertia of the dummy member is equal to the column inertia in the case of a rigid base, and a value of 10% of the column inertia in a nominally pinned base. In the case of portal frames, it is permissible to model the base stiffness as 10% of the column stiffness for the ultimate limit state, and 20% of the column stiffness for the serviceability limit state[18]. If this procedure is adopted, it is probable that separate analyses will be required for both limit states.

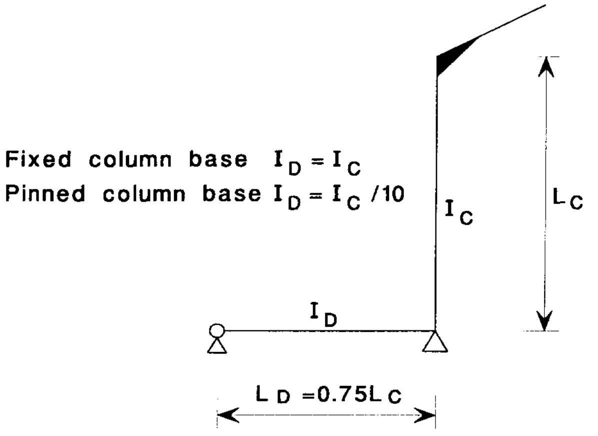


Figure 38 *Alternative model of base stiffness*

Many programs permit base stiffness to be input directly as a spring stiffness. In this case a rigid base is input as $4EI_c/L_c$ and a nominally pinned base as $4EI_c/10L_c$.

The column base connection to the foundation is another grey area, both in modelling and reality, and the distinction between pinned and fixed bases can be difficult to define in practical details. Portal frames are usually analysed with pinned bases, since the cost of moment resisting foundations often exceeds the savings in frame weight achieved by using fixed bases. It is uncommon, however, to see details which are immediately recognised as pinned (Figure 39). More common are details shown in Figure 40 which are frequently deemed to be pinned in analysis. Details such as these are preferred for two reasons:

- The use of four holding down bolts allows the column to be erected without guying or propping, and permits easier adjustment and plumbing.
- A moment resisting base may be required for stability during fire, if the column is situated near a site boundary. The Building Regulations define when a building must incorporate this requirement. The reader is referred to *The behaviour of steel portal frames in boundary conditions*[19].

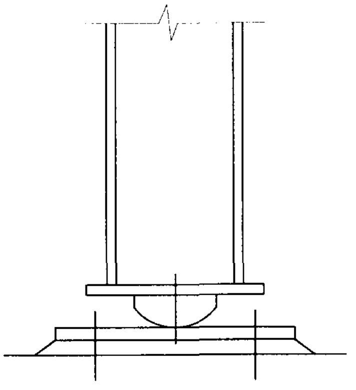


Figure 39 *Base with bearing*

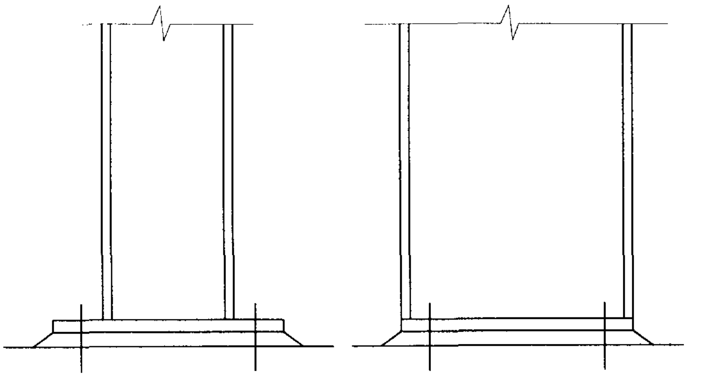


Figure 40 *Typical pinned base details*

It will be noted, however, that the bases shown in Figure 40 could also be classed as moment resisting. In the case of boundary columns, the base detail must be capable of resisting moment, although such bases are generally modelled as pinned for the frame analysis.

11.2 Horizontal and vertical fixity

Rotational base fixity was discussed in Section 11.1, but most programs also allow vertical and horizontal support options of rigid, free and a spring stiffness. The reader's attention is again drawn to Reference 16 for detailed advice on the subject.

Differential settlement is usually more damaging than overall settlement, although detailed advice is difficult. Often ignored, settlement of isolated foundations can have a dramatic effect on the bending moments on the frame. Figure 41 shows a typical bending moment diagram provided as a result of the third foundation being displaced vertically by 30 mm, relative to the remainder.

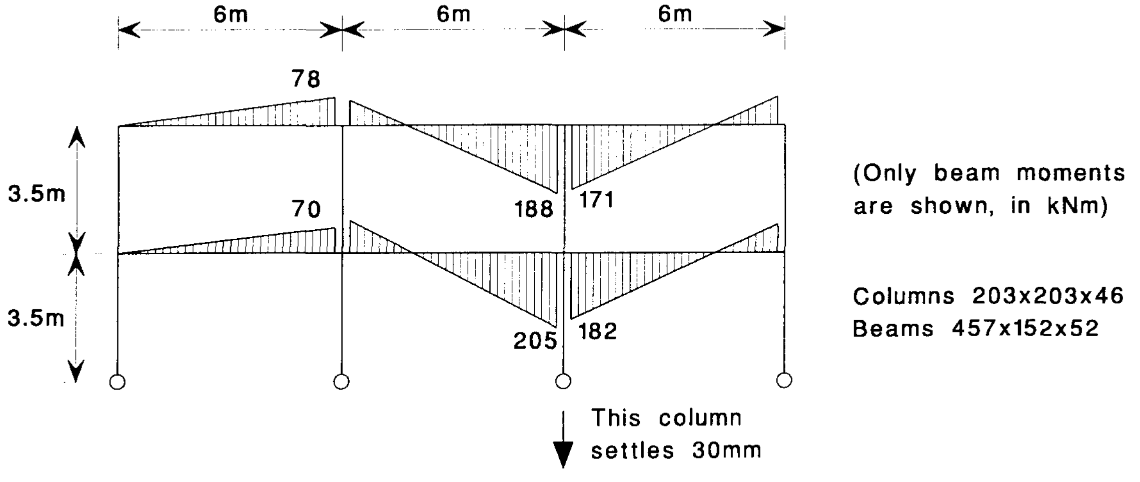


Figure 41 *Example of bending moments due to foundation settlement*

If foundation details and soil properties are known, spring supports can be introduced to model the compressibility of the soil.

Horizontal releases are frequently necessary if the model is to properly reflect reality. Apart from the foundations, any other support is almost certain to allow the structure to 'spread', and one or more supports must be released to reflect this.

To illustrate this point, consider a triangulated roof truss, simply supported by two columns in Figure 42. If designing the truss in isolation, the supports must be modelled with a horizontal release, or the analysis will produce compression in some panels of the bottom boom - clearly incorrect in this situation.

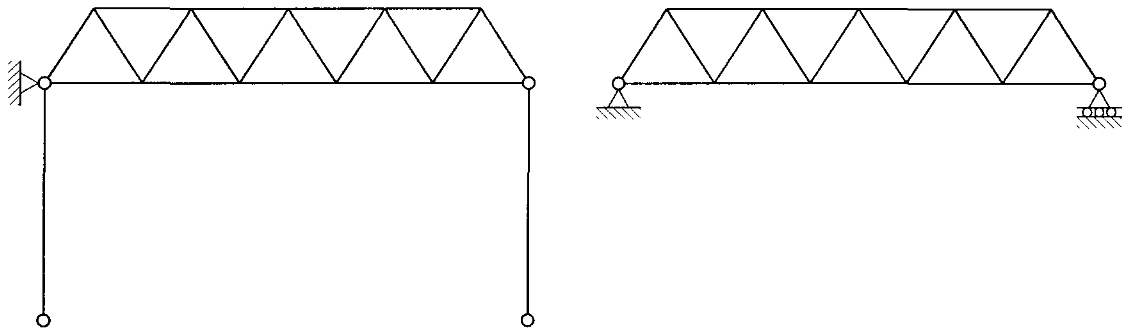


Figure 42 *Typical roof truss - reality and analysis model*

11.3 Recommendations

If a structure is analysed with pinned bases, but in reality the bases are semi-rigid, the bending moments produced by the analysis are generally conservative. Analysis using pinned bases is therefore recommended, even if the base details appear capable of resisting moment.

Fixed bases should not be specified without a consideration of the effect of the fixity on the foundation costs, which can become prohibitively expensive. The structural designer should note that the provisions of Clause 5.1.2.4 preclude full fixity being assumed in the analysis.

The capacity of most nominally pinned bases to resist moment (as acknowledged in Clause 5.1.2.4(b)) may be used to advantage, particularly in the reduction of sway deflections.

Without detailed investigation of ground conditions and foundation behaviour, it is generally acceptable to assume the foundation supports to be rigid vertically and horizontally, acknowledging the ability of a steel frame to redistribute moment and behave in a ductile manner.

12 MODELLING OF LOADS

12.1 General

In most structures, the magnitude of loads cannot be determined precisely, and the loads used in analysis represent an estimate of the likely maximum load to which the structure will be exposed. Some loads, such as the self weight of a structure, may appear easier to estimate than others, such as wind loads. The estimate of imposed loads such as wind and snow can be based on observation of previous conditions and the application of a probabilistic approach to predict maximum effects which might occur within the design life of the structure.

Loads associated with the use of the structure, such as imposed floor loads, can only be estimated based on nature of usage. Insufficient data is available in most cases for a fully statistical approach and notional values are therefore assigned by national standards.

In limit state design, characteristic values of load are used as the basis of all design. They are values which statistically have only a small probability of being exceeded during the life of a structure. To provide a margin of safety, particularly against collapse, partial safety factors are applied to these characteristic values to obtain design loads. In principle, different partial safety factors can be applied depending on the degree of uncertainty or variability of a particular type of load. In practice, whilst this appears to be the case, the actual values of partial safety factors used incorporate significant elements of the global safety factor and do not represent a rigorous probabilistic treatment of the uncertainties of the actions.

12.2 Modelling of loads

Once the loads to be taken into account have been identified, the application of the loads to the analysis model will depend largely on the degree of simplification present in the analysis model. A three dimensional model including secondary elements is likely to have a complex application of loads, compared with a plane frame analysis where the characteristic loads will be further simplified. Considering a portal frame, the wind load and roof load will be applied to the main frames via secondary elements such as purlins or sheeting rails, which in turn are loaded via the cladding. If the purlins or rails are included in a three dimensional model, the load should be applied to these elements. If single two dimensional frames are modelled, the equivalent point loads should be calculated at purlin positions (if known at that stage), or an equivalent uniformly distributed load may be applied to the frame.

Simplification in calculating equivalent point loads and distributed loads is recommended. In the floor slab shown in Figure 43, the floor beams will usually be designed for a uniformly distributed load as shown in (a), not the distribution shown in (b).

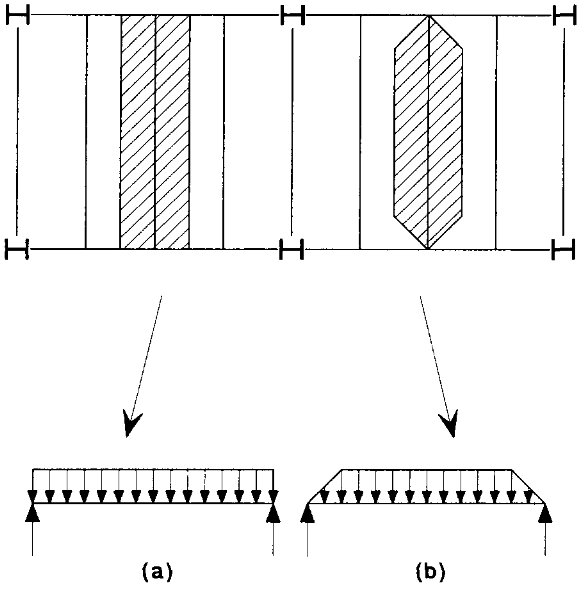


Figure 43 *Simple modelling of loads*

Similarly, multiple point loads on any member may be treated as a distributed load. Five or more equally spaced identical point loads (Figure 44) on a member may generally be considered as a distributed load, without significant loss in the accuracy of the analysis. *Steel Designers' Manual*[8] illustrates the effects of multiple point loads and of alternative point load distributions.

5 or more equal loads ≡ uniformly distributed load

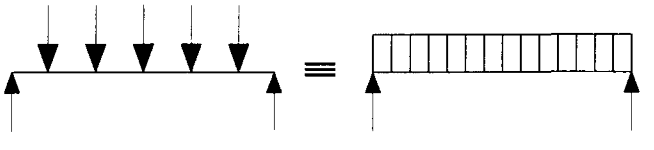


Figure 44 *Equivalent uniformly distributed load*

12.3 Load types

General analysis programs will have a range of ways in which the model may be loaded. Loading will require definition relative to either global or local (member) axes. The following list is not comprehensive, but indicates the options that are usually available.

- Joint loads
- Joint displacements
- Uniformly distributed loads on elements
- Varying distributed loads on elements
- Point loads on elements
- Self weight
- Temperature change
- Member distortions.

Some of these options may be limited, depending on the type of structure, and the type of analysis. Truss models, for example, may preclude distributed load on elements, and only allow loads to be input at nodes.

12.4 Load combinations

As most general analysis packages will allow the creation of factored combination load cases using the principle of superposition, the structural designer will find advantage in describing each basic load case separately, using unfactored loads. Combinations of basic load cases can then be created, applying the appropriate load factors in each combination. This facilitates the simple composition of ultimate load combinations, and if required, suitable serviceability load combinations to determine deflections. Advantages of this approach are:

- In design software, the resistance checks will be made against the ultimate load combinations, and the serviceability checks made against the appropriate unfactored load cases, or serviceability combinations.
- Unfactored reactions are readily obtained for foundation design.
- Notional load cases may be readily created.

12.5 Recommendations

Calculation of equivalent loads to be applied to the model should be simple, and err towards conservatism, noting the uncertainty of the characteristic loads, and that any modelling of loads is an approximation to the loads on the real structure.

When five or more equal point loads equally spaced apply to a single member, a uniformly distributed load may be modelled.

Unfactored basic load cases should be entered, and factored combinations created using the principle of superposition. This principle cannot be used in plastic, elastic-plastic or second-order analyses.

13 INPUT AND OUTPUT CHECKS

Probably the most important part in any analysis exercise is to review the output in order to confirm that an appropriate structural model has been used, and that the applied loads are correct. This is **not** to confirm that the execution of the analysis is correct! When using proven software the analysis will be correct - the exercise is to check the structural designer's input.

Software frequently contains default values for certain input data. Support fixity and restraint conditions are common examples of data which may have a default value. Default values are intended to avoid the necessity for the structural designer to enter data, and represent the 'usual' condition, which may be amended by the structural designer. The structural designer must give due regard to the default values assumed by the program, and either satisfy himself that these are appropriate, or amend the value accordingly. All input data, whether default or user input, remains the responsibility of the structural designer. Default values are common in both analysis and design software.

Program Defaults

Are these correct? Are the default conditions appropriate for the physical details and the loading of the frame?

Loading

Viewed graphically, do the loads in each loadcase appear correct? Are any elements without load? Do loads on some elements appear to be orders of magnitude different from others? Are the loads applied in the correct orientation?

Deflected Form

Is the deflected form correct? Has the structure deflected as expected, and is the **order** of the overall deflection as expected?

Bending Moment Diagram

Is the form of the bending moment diagram as expected? Are moments shown where releases would have been appropriate? Does the overall envelope on a member equate to that calculated by a simplistic approach, typically $wL^2/8$ or $WL/4$?

Reactions

Checking by hand calculation, do the total reactions provided in the output (vertically and horizontally) equate to the applied loads? Do the reactions quoted for different unfactored loadcases differ by orders of magnitude? Is the distribution of load to the supports as expected?

Spring Stiffnesses

Are the spring stiffness values assumed for analysis appropriate to the members as designed?

Trusses

If the truss is simply supported and carrying a uniformly distributed load, do the maximum chord forces equate to $wL^2/8$, divided by the depth of the truss?
Does the vertical component of the end internal member equate to the vertical end reaction?

Are the displacements of the correct form and order? In the analysis model, is one end of the truss free to move longitudinally?

Do redundant members at the ends of the truss (if included in the model) attract load, or have they been released?

14 MEMBER DESIGN

14.1 Introduction

In a guide to analysis of steel structures, a section on member design may appear out of place, but is readily justified by the frequent use of software design packages as an immediate follow on from analysis. Many compatible analysis and design suites are in common use within the industry, and to move between analysis and design is generally so seamless that the separate processes are considered as one in the minds of many structural designers.

The purpose of this Section is not to provide a comprehensive treatise on computer aided design, but in the same spirit as the earlier Sections, to provide some guidance and warnings for the unwary or inexperienced designer.

14.2 Restraint to buckling

If a facility to restrain a member is provided, it is likely that the program considers the restraints effective in all load cases. In reality, this is unlikely. Consider a simply supported rafter subject to both gravity load and uplift load cases: all purlins provide restraint in the gravity load case, but only those with stays to the bottom flange do so in the uplift case. Two separate design runs may therefore be necessary to check both cases, with different restraint conditions. The nature of the restraint assumed by the module must be investigated. Some restraints (for example the bottom flange restraint of a portal rafter) restrain against buckling in the y-y direction and against lateral torsional buckling. Buckling about the x-x axis is not affected. The introduction of a restraint **may** assume by default an effective restraint against lateral torsional buckling, and buckling in **both** axes. Programs usually have the option to re-introduce the correct buckling length in the appropriate direction.

14.3 Serviceability checks

Default deflection checks are likely to be included, which may need revision. In most general analysis and design software, beams and columns will be checked for deflection within their own length; only cantilevers will be checked at the tip. The structural designer may be tempted to use this facility when checking the sway of the frame, expecting that the deflection quoted for the column will be the deflection, Δh, measured at the column top. The quoted deflection is likely to be that calculated within the length of the column, δ, as shown in Figure 45. It may be convenient to re-classify the columns as cantilevers in order to apply the correct checks, or in order to base the member design on the deflection of the column top.

For manual checking, displacements at nodes will generally be quoted as global displacements, and not relative to particular elements in the model.

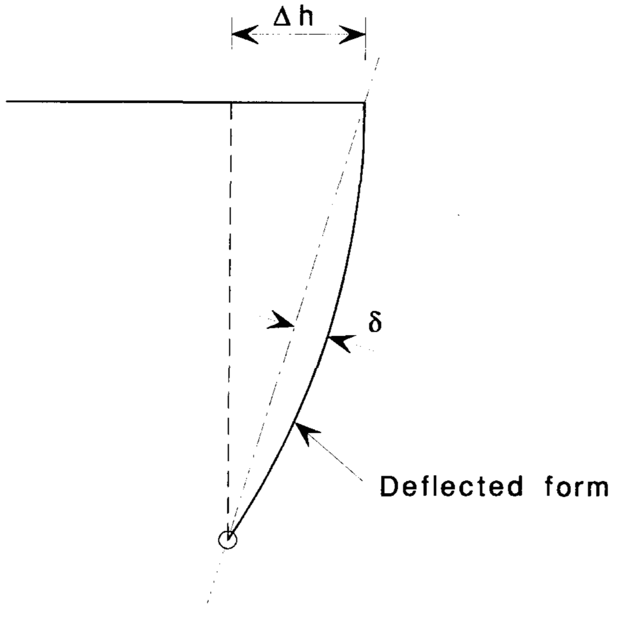


Figure 45 *Deflection checks of column member*

Caution must also be exercised when checking deflection of a series of elements connected longitudinally, for example in a truss chord. The quoted deflection may well be of an individual element between nodes; the actual joint deflections from the analysis output must be considered in order to check the **overall** deflection. Some programs have a facility to check the overall deflection by identifying the elements to consider as one single member.

The deflections resulting from the analysis will be based on the initial section properties, and whilst the design module may perform a pro-rata adjustment when calculating the deflection, a re-analysis with the chosen sections will be beneficial if the deflections are close to the allowable limit. This may be particularly significant in rigid frames, where re-sizing the elements will affect the distribution of moments and the deflection of the structure.

14.4 Effective lengths

These will usually be given a default value in the x-x and y-y directions, to be modified by the structural designer in accordance with the appropriate code Clause (4.3.5, 4.3.6, 4.7.2 and 4.7.10).

The effective lengths assumed by the program will be based on the length of the element. Considering a truss for an example, the effective length of the top (compression boom) may be related to the node positions in one direction, and the purlin positions in the other. More importantly the bottom boom in reversal is likely to have an effective length between restraint positions, usually considerably more than the distance between nodes. The structural designer must therefore review and amend as necessary the effective lengths assumed by the program.

The introduction of restraints within the length of an element requires a similar review of the revised effective lengths in each direction.

14.5 Destabilising loads

A conservative default condition to destabilising loads is usual, with the option to change. This should be checked by the structural designer.

14.6 Minimum weight

Many design programs have a minimum weight design option, which produces the lightest section satisfying code requirements. This is a useful option, and saves the trial design of different sections, and certainly there is no value in providing excessive capacity compared to the imposed forces and moment. However, it must be appreciated that a least weight solution is generally not the cheapest solution overall. The form of the relationship between cost and weight is indicated in Figure 46.

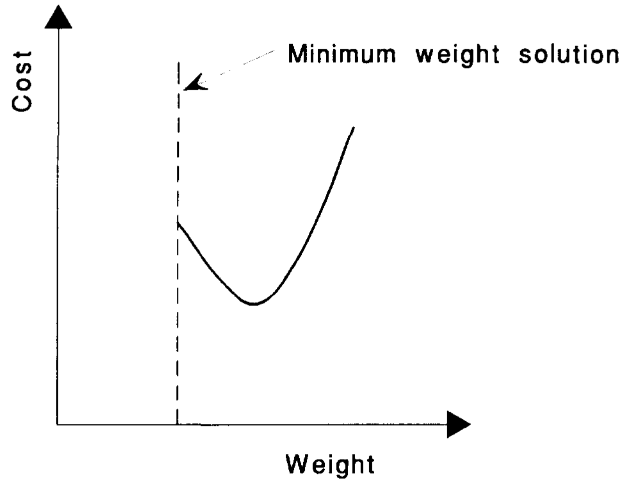


Figure 46 *Relationship between cost and weight*

As a minimum weight solution is approached, fabrication costs increase dramatically, mainly due to the need for local stiffening at the connections. Fabrication costs also increase as standardisation and repetition decrease. Guidance on cost and connection capacities can be found in *Design for Manufacture Guidelines*[(2)].

In addition to local stiffening, a least weight solution can lead to members which become increasingly impractical to connect, simply due to physical size. As a general guide, members with flanges too narrow for 20 mm bolts should be avoided. Connections to the webs of shallow sections are best avoided, particularly if members are also connected to the flanges. This can cause difficulty on site, due to the congestion in the connection area, in addition to the necessity to notch the members connected to the web. As a typical example, whilst a 152 UC column might be chosen to support four floor beams, a 203 × 133 beam (used as a column) may also satisfy the design requirements, but provide better access, with no notching of beams. (Figure 47).

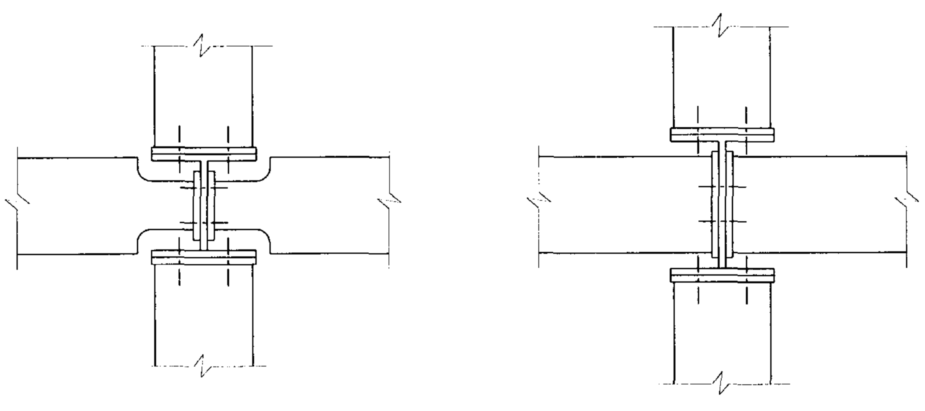


Figure 47 *Comparison of connection detail to small column*

Section sizes should be rationalised where possible. In particular, the same steel section should be chosen for each element of a member (for example a truss chord).

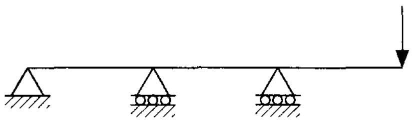
14.7 Design options

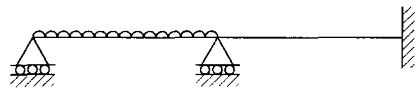
Design suites frequently have a range of additional facilities to those already described, including:

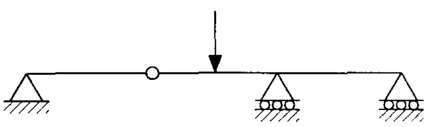
- Minimum depth design to produce the shallowest section satisfying code requirements.
- The option to choose which load cases will be included in which checks. The ultimate load case combinations should be used for strength checks, and appropriate unfactored load case combinations for serviceability checks.
- The facility to generate nominal moments due to eccentric connections to columns in 'simple' construction, and to include these in design.
- The facility to change generic member type, for example from universal beams to universal columns or hollow sections. This allows swift comparison between suitable members chosen from the various types.
- The facility to re-analyse the structure based on the sections chosen in the most recent design. If the designed sections differ widely from those used to build the analysis model, the changes in forces, moments and deflections can be significant. The software may (or may not) provide a warning that the members in the current design differ from those used in the analysis.

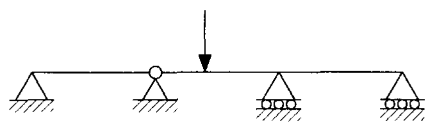
APPENDIX A: Self assessment exercise

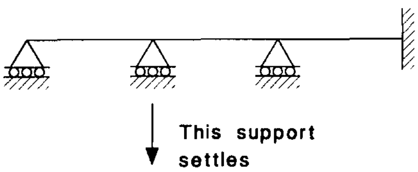
The following frames formed part of an exercise[5] to measure the ability of graduate engineers to assess qualitatively the behaviour of simple frames. In each example, the bending moment diagram should be sketched. Answers are not provided!

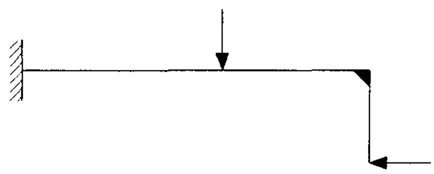
1


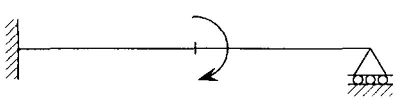
2


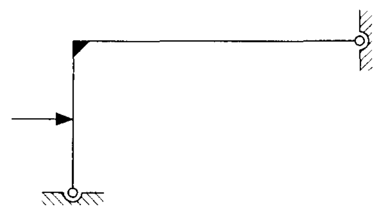
3


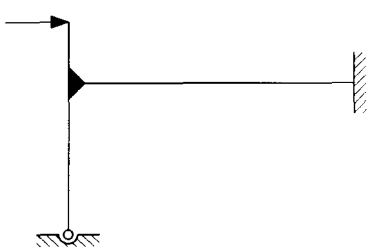
4


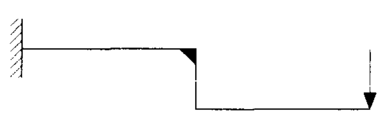
5


6


7


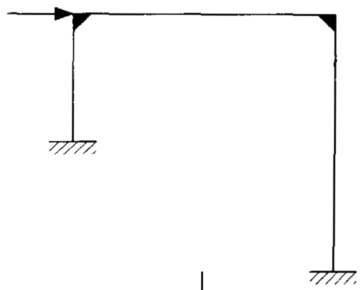
8


9


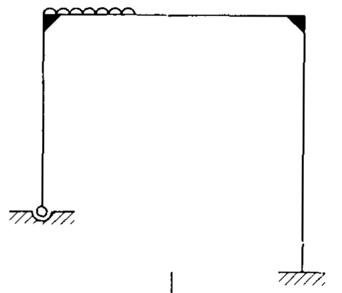
10


(continued)

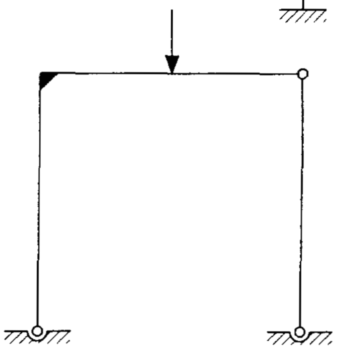
11



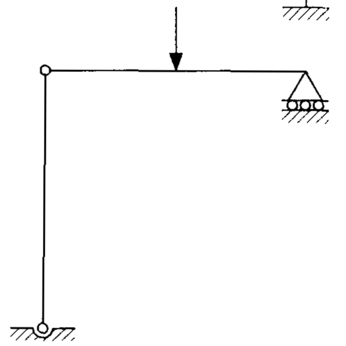
12



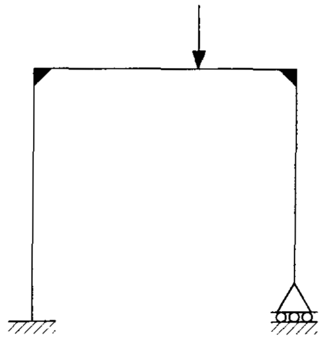
13



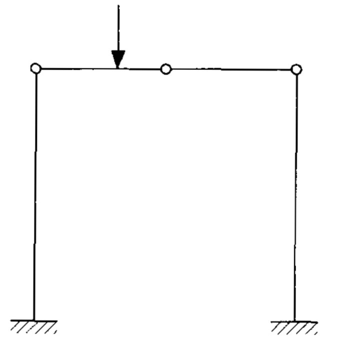
14



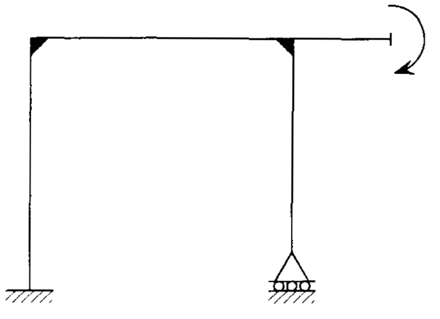
15



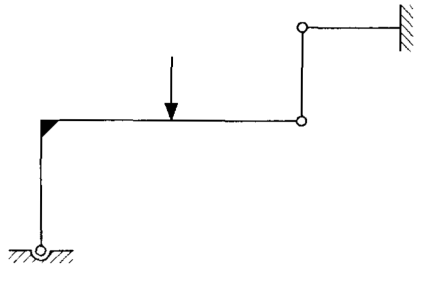
16



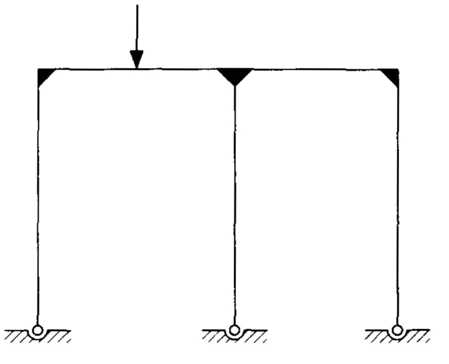
17



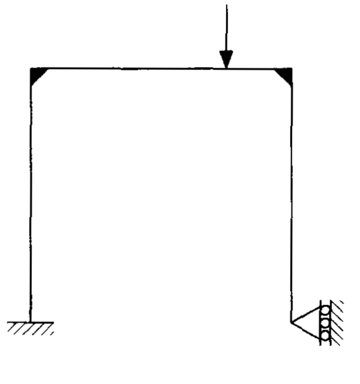
18



19



20



APPENDIX B: P-delta effects

P-delta effects are additional displacements forces and moments which arise from the action of the applied loads on a deflecting structure. They are second-order effects and can be illustrated by the example shown in Figure B1.

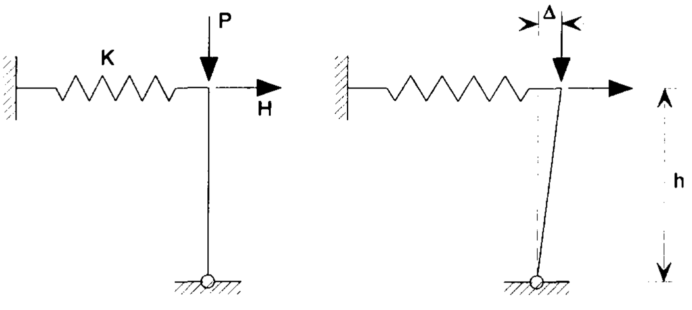


Figure B1 *P-delta effect*

A vertical strut with the bottom end pinned and the top end restrained by a spring is subject to a compressive axial load and a horizontal load at the top. (This is equivalent to a vertical cantilever subject to the same loading if the spring stiffness is made equal to the flexural stiffness of a cantilever under a point load at its tip).

Taking moments about the pin,

$M_1 = Hh$ (first-order effect)

$M_2 = P\Delta$ (second-order effect)

(hence the name P-delta)

For equilibrium, the force in spring

$$F = (M_1 + M_2)/h$$

$$= H + P\Delta/h$$

The additional force in the spring ($P\Delta/h$) is due to the P-delta effect. The total force in the spring is also equal to the spring stiffness multiplied by the displacement, so that:

$$K\Delta = H + P\Delta/h$$

or, rearranging,

$$H = \Delta(K - P/h).$$

The modified lateral stiffness (H/Δ), taking into account the second-order effects is therefore ($K - P/h$) and is clearly reduced by the action of the vertical load.

If the second-order moment $M_2 = P\Delta$ is very small with respect to the first-order moment, the P-delta effect can be ignored. Clause 5.1.3 gives guidance when this is permitted. The guidance is expressed in terms of the deflection over a single storey height, which is equivalent to the quantity Δ/h in the above example.

APPENDIX C: Notional horizontal forces

Notional horizontal forces are horizontal forces which are related to the vertical load on a building (they are not 'real' horizontal loads). They have three distinct functions in BS 5950: Part 1. These are:

(a) To ensure that a structure has adequate strength against sway (necessary when design applied lateral loads are small). Here the notional horizontal forces provide a minimum level of lateral load for design. (Clause 2.4.2.3)

(b) They allow for the possible vertical imperfection of the structure. (Clause 2.4.2.3)

(c) They can also be used to determine approximately the increase in load effects in a building due to P-delta effects, where a linear-elastic analysis is carried out. The method by which this is done is known as the amplified sway method. (Clause 5.6.3 and Appendix F).

C.1 Amplified sway method

The amplified sway method is a way of estimating P-delta effects by first-order linear elastic analysis only. The application of the method may be illustrated by using the example in Appendix B, as follows.

If the horizontal force H is related to the vertical load P by the relationship:

$$H = \alpha P$$

and this load is applied to the model in the absence of vertical load, the spring will extend by an amount δ. The spring stiffness is therefore given by:

$$K = \alpha P/\delta.$$

If in the absence of any horizontal load the vertical load P is increased to P_{crit} , when the strut is on the point of collapse, for any given value of displacement Δ, there will just be equilibrium between the overturning moment from the vertical load and the restoring moment from the spring, i.e.:

$$P_{crit}\Delta = K\Delta h$$

Substituting for K and eliminating Δ,

$$P_{crit} = (\alpha P/\delta)h.$$

This can be written:

$$(P_{crit}/P) = \alpha(h/\delta)$$

The quantity (P_{crit}/P) is known as the elastic critical load factor which is given the symbol λ_{cr} in BS 5950: Part 1. The quantity (h/δ) is the reciprocal of the sway index, ϕ_s. If the magnitude of the notional horizontal forces used to determine the spring stiffness is taken as 0.5% of the vertical load, the formula for elastic critical load factor is the same as that given in Appendix F of BS 5950: Part 1.

C.2 Amplified sway method - worked example

The multi-storey rigid frame shown in Figure C1 is to be analysed elastically.

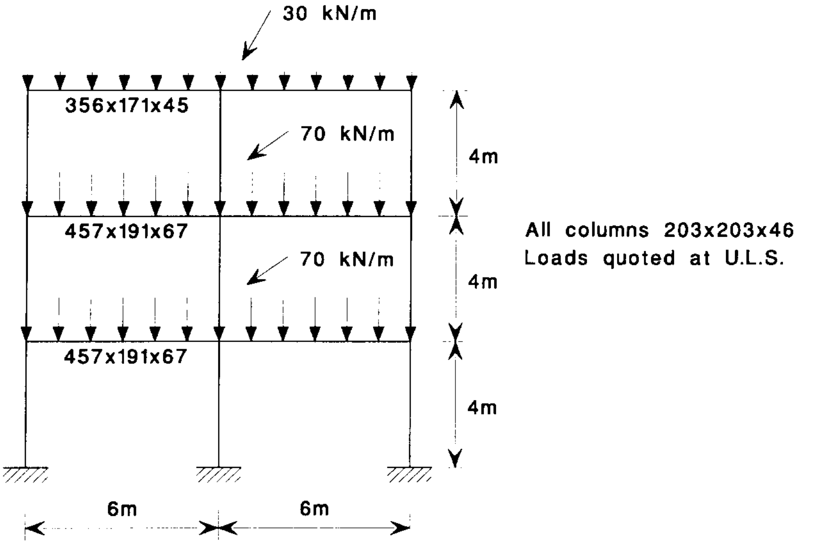


Figure C1 *Example frame for analysis*

Clause 5.6 refers the structural designer to Clauses 5.1 and 5.2. Clause 5.1.2.5(a) covers the stiffness of fixed bases, and allows the foundation stiffness to be taken as equal to that of the column. Following the advice given in Section 11 of this publication, the structure is modelled as shown in Figure C2, with the length of the dummy member as $0.75 \times 4m = 3m$.

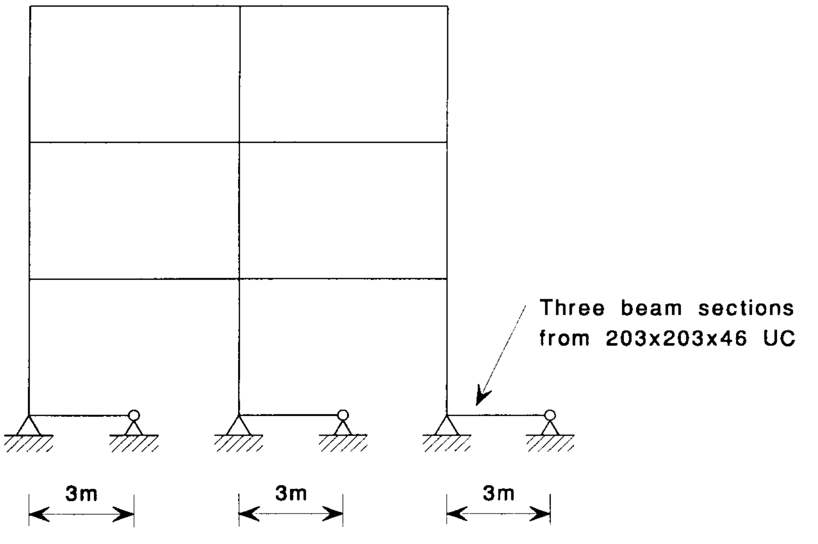


Figure C2 *Frame model used in analysis*

Clause 5.1.3 requires the frame to be checked and classed as sway or non-sway, by determining the storey deflections due to the notional loading in Clause 5.1.2.3.

From 5.1.2.3, the notional horizontal loads are:

at first and second floors: $\dfrac{70 \times 12 \times 0.5}{100} = 4.2$ kN

at top floor: $\dfrac{30 \times 12 \times 0.5}{100} = 1.8$ kN

The notional horizontal loading, and resulting lateral deflections are shown in Figure C3.

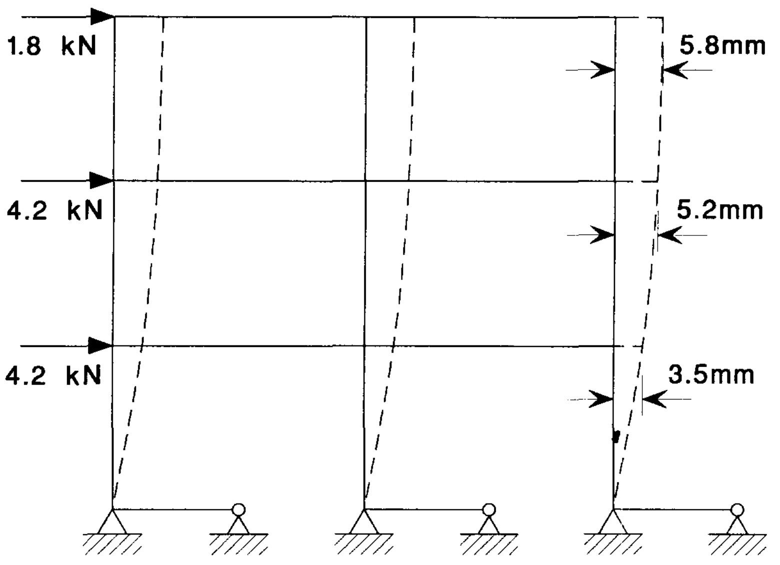


Figure C3 *Lateral deflections due to notional horizontal loads*

By inspection, the deflection from ground to first floor (3.5 mm) is critical.

The limiting deflection given in Clause 5.1.3 for a non-sway frame (ignoring the stiffening effect of cladding) is:

$$\frac{h}{2000} = \frac{4000}{2000} = 2 \text{ mm}$$

The frame must therefore be classified as a sway frame, and the provisions of Clause 5.6.3 apply.

Clause 5.6.3 allows two methods to be used to account for the effect of sway; by Extended Simple design or by the Amplified Sway method. The latter is suited for computer analysis, and firstly involves calculating the elastic critical load factor, λ_{cr}, from Appendix F.

Appendix F, Clause F.2.1, requires an ordinary linear elastic analysis to determine the horizontal deflections of the frame due to the notional horizontal loads, with allowance made for the degree of rigidity in the base in accordance with Clause 5.1.2.4.

This analysis has in fact already been completed, when making the classification of the frame as a sway frame. The largest sway index, ϕ_s, of any storey, is between the ground and first floor, and in accordance with Clause F.2.4 is:

$$\phi_s = \frac{3.5}{4000} = 8.75 \times 10^{-4}$$

then λ_{cr} (Clause F.2.3) $= \dfrac{1}{200 \times 8.75 \times 10^{-4}} = 5.71$

Clause 5.6.3(b) requires the moments due to the actual horizontal loading to be amplified by the factor:

$$\frac{\lambda_{cr}}{(\lambda_{cr} - 1)} = \frac{5.71}{(5.71 - 1)} = 1.21$$

This amplification factor can be introduced when creating the loadcase combinations.

REFERENCES

1. STANDING COMMITTEE ON STRUCTURAL SAFETY
Tenth Report
SETO Ltd, 1994

2. CIMsteel
Design for manufacture guidelines
The Steel Construction Institute, 1995

3. BRITISH STANDARDS INSTITUTION
BS 5950: Structural use of steelwork in building
Part 1: Code of practice for design in single and continuous construction: hot rolled sections
BSI, 1990

4. NERVI, P. L.
Structures
F. W. Dodge Corporation, 1956

5. BROHN, D. M., COWAN, J.
Teaching towards an improved understanding of structural behaviour
The Structural Engineer, Vol. 55, No 1, 1977

6. BROHN, D. M., COWAN, J.
Discussion on the paper 'Teaching towards an improved understanding of structural behaviour'.
The Structural Engineer, Vol. 55, No 11, 1977

7. Verulam
The Structural Engineer, Vol. 70, No 22, 1992

8. THE STEEL CONSTRUCTION INSTITUTE
Steel Designers' Manual - 5th Edition
Edited by Owens, G. W., and Knowles, P. R.
Blackwell Scientific Publications, 1992

9. LAWSON, R. M.
Commentary on BS 5950: Part 3: Section 3.1, 'Composite Beams'
The Steel Construction Institute, 1990

10. CIDECT
Construction with hollow steel sections (Various titles)
(Available via British Steel Tubes and Pipes, Corby.)

11. BRITISH STANDARDS INSTITUTION
DD ENV 1993-1-1:1992 Eurocode 3: Design of steel structures
Part 1.1. General Rules and rules for buildings (together with United Kingdom National Application Document).
BSI, 1993

12. BRITISH STEEL
Design of SHS welded joints (and associated guides)
Available from British Steel Tubes and Pipes, Corby

13. THE STEEL CONSTRUCTION INSTITUTE &
THE BRITISH CONSTRUCTIONAL STEELWORK ASSOCIATION
Joints in simple construction
Volume 2: Practical applications
BCSA/SCI, 1992

14. LI, T.Q., CHOO, B.S. and NETHERCOT, D.A.
Connection element method for the analysis of semi-rigid frames
Journal of constructional steel research, Vol 32, 1995

15. THE STEEL CONSTRUCTION INSTITUTE &
THE BRITISH CONSTRUCTIONAL STEELWORK ASSOCIATION
Joints in steel construction: Moment connections
SCI/BCSA, 1995

16. INSTITUTION OF STRUCTURAL ENGINEERS
Soil - Structure Interaction. The Real Behaviour of Structures
Institution of Structural Engineers, 1989

17. THE STEEL CONSTRUCTION INSTITUTE
Advisory Desk AD 097
in Steel Construction Today, Vol. 5, No.6
SCI, 1991

18. THE STEEL CONSTRUCTION INSTITUTE
Advisory Desk AD 090
in Steel Construction Today, Vol. 5, No.4
SCI, 1991

19. NEWMAN, G. M.
Fire and steel construction - The behaviour of steel portal frames in boundary conditions, 2nd Edition
The Steel Construction Institute, 1990

BIBLIOGRAPHY

GREINER, R., SALZGEBER, G., PIRCHER, H., and HANSEN, J. R.
A structured approach to design, analysis and code check of constructional steelwork
(Draft document)

BROHN, D. M.
Understanding structural analysis, 2nd Edition
Blackwell Scientific Publications, 1990

INSTITUTION OF STRUCTURAL ENGINEERS
Qualitative analysis of structures
Institution of Structural Engineers, 1989

BROHN, D. M.
Structural engineering - a change in philosophy
The Structural Engineer, Vol. 60A, No. 4, 1982

DAVIES, J. M.
In-plane stability of portal frames
The Structural Engineer, Vol. 68, No. 8, 1990

DAVIES, J. M.
The stability of multibay portal frames
The Structural Engineer, Vol. 69, No. 12, 1991