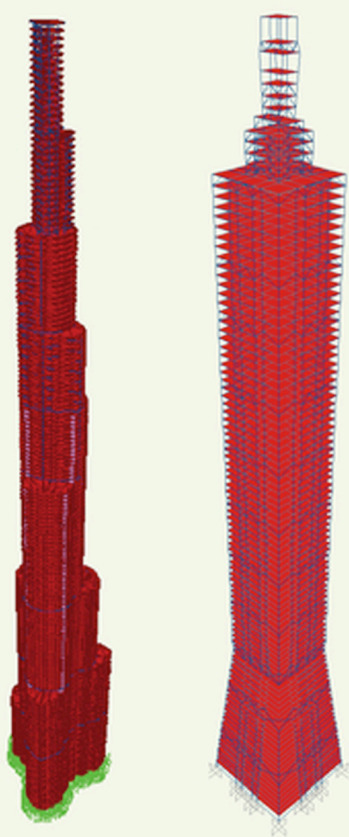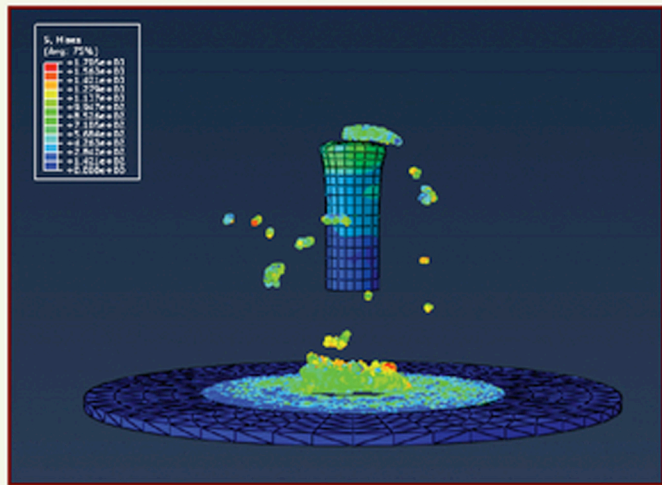# SET SA

ارائۀ به‌روزترین منابع، کتاب‌ها و جزوات مهندسی عمران

به زبان فارسی و انگلیسی به‌صورت کاملاً رایگان

www.TheSetosa.com

ADVANCED MODELLING
TECHNIQUES IN
STRUCTURAL DESIGN

FENG FU

WILEY Blackwell

Advanced Modelling Techniques in Structural Design

Advanced Modelling Techniques in Structural Design

Feng Fu

City University London

WILEY Blackwell

1   2015

Contents

About the Author

Dr Feng Fu received his PhD from the University of Leeds and MBA from the University of Manchester. He is a Chartered Structural Engineer and Member of American Society of Civil Engineering. He is currently a Lecturer in Structural Engineering in City University London following his work at the University of Bradford in the same position.

Prior to that, he worked for several world-leading consultancy companies and was involved in the design of several prestigious construction projects worldwide. He worked in the advanced analysis team in the WSP Group Ltd London, following his work as a Structural Engineer in the Waterman Group Ltd London. Prior to commencing his PhD in the UK, he also worked as Structural Engineer for one of the best design companies in China, the Beijing Institute of Architectural Design and Research.

During his industrial practice, he worked with several world-leading architects such as Zaha Hadid, Forster and SOM. He has designed and analysed all kinds of complex and challenging structures, such as tall buildings, long-span space structures and bridges. He has also gained extensive experience in designing buildings under extreme loadings, such as blast and fire, and designing buildings to prevent progressive collapse.

Dr Fu has extensive research experience in the areas of progressive collapse, buildings under extreme loadings such as blast and fire, Tensegrity structures and composite joints. He has specialised in advanced numerical modelling and has developed several modelling programs using different languages such as FORTRAN and Visual Basic. He has also carried out several full-scale tests on composite joints. His recent research has focused on investigating the behaviour of high-rise buildings, bridges and offshore structures under extreme loads such as blast and fire using advanced 3D numerical modelling techniques. He has published a number of refereed journal papers as the first author and is also a reviewer for over 15 international journals and two books.

Preface

Analysis of complex structures has become increasingly important and impressive progress has been made over the last two decades. Thanks to the advent of computers and the development of different numerical modelling methods, engineers are capable of designing more challenging buildings, such as Buji Khalifa, Taipei 101, Millau Viaduct and so on.

I have worked in both the industry and academia for many years and have noticed that many engineers lack knowledge of the theories and modelling techniques in analysis of complex structures, as well as some special design problems such as vibration, fire, blast and progressive collapse. There is also a large knowledge gap for students, in addition to which most have difficulty designing and analysing real construction projects.

The motivation behind this book is to provide engineers with an understanding of the featured design problems for different types of structures, with an effective way to model these types of structures using conventional commercial software and with the theories and design principles that underpin the relevant analysis.

While I worked in the advanced analysis team in the WSP group, I gained experience in different kinds of structural analysis problems and modelled many complex structures, from tall buildings to long-span structures. I worked out many methods to effectively model them, using just continental analysis programs, and I feel it is necessary to share these methods with readers.

While teaching at the University of Bradford and City University London, I started to teach my final year and Masters students how to model existing complex buildings around the world in their graduation projects, such as Buji Khalifa and the Millau Viaduct. It is great to see that these have become the most popular projects. The students learnt both design principles and modelling methods through these projects. Therefore, another objective of this book is to provide civil engineer students with detailed knowledge in design and analysis of complex structures.

Thus, this book has been written to serve not only as a textbook for college and university students, but also as a reference book for practising engineers. This book covers almost all the structural design problems an engineer may face, such as lateral stability analysis for tall buildings, earthquake analysis, progressive collapse analysis, structural fire analysis, blast analysis, vibration analysis, nonlinear geometric analysis and buckling analysis. Another feature of this book is that most of these analysis methods are demonstrated using existing prestigious projects around the world, such as Buji Khalifa, the Willis Towers, Taipei 101, the Gherkin, the Millennium Bridge, the Millau Viaduct, the Forth Bridge and so on. This is to help develop understanding of

effective ways to model complex structures. In addition, this book also introduces the latest Building Information Modelling system, which is a new way forward in design and analysis of modern projects. The features of major commercial programs used in the industry are also introduced, which provides guidance for readers on the selection of analysis programs.

Feng Fu

Acknowledgements

1 Introduction

1.1 Aims and scope

With the fast development of modern construction technology, major international city skylines are changing dramatically. More and more complex buildings, such as Burj Khalifa in Dubai, the Birds Nest Stadium in Beijing and the London Aquatic Centre, have been built over the past decade. As a Chartered Structural Engineer, the author has worked for several leading international consultancy companies and has worked on several prestigious projects around the world. The experience of the author demonstrates that in current design practice most of these buildings could not have been designed without the use of advanced modelling techniques. Fierce competition in the current design market also requires structural engineers to handle the increasing difficulty in designing the more complicated projects required by both clients and architects. This challenge can only be tackled by using modern computer technology. It also imposes a big change in the role of the structural engineer: in addition to knowledge of basic design principles and structural analysis methods, an engineer should also have a full understanding of the latest modelling techniques. This is also the reason that advanced computer modelling skills have recently become essential for an engineer's recruitment by increasing numbers of design consultancies.

However, in the construction industry, most structural engineers find themselves lacking modelling knowledge, as few textbooks have been provided in this area. For students, although some elementary modelling techniques are taught in most Civil Engineering courses, no systematic introduction is made, let alone how to model a real construction project in practice. Therefore, a book in this area is imperative.

The main purpose of this book is to introduce and provide detailed knowledge of advanced numerical analysis methods and important design principles for both students and design practitioners. It addresses effective modelling techniques in solving real design problems and covers a broad range of design issues – such as lateral stability of tall buildings, buckling analysis of long-span structures and earthquake design – and some special issues such as progressive collapse, blast, structural fire analysis, foot-induced vibrations and so on.

It also introduces a variety of major modelling programs (such as SAP2000, ETABS, Abaqus®[1], ANSYS) and preprocessing software (Rhino, Revit, AutoCAD) used in current structural design practice. A number of modelling examples using this software

---

[1] Abaqus® is a registered trademark of Dassault Systèmes and/or its subsidiaries.

are provided in the book. Most of the model examples are based on a worldwide selection of real design projects, such as the Millennium Bridge and Burj Kalifa, helping readers to find an effective way to model these types of structures.

In addition, the algorithms and theories that underpin the analysis, such as the finite element method (FEM) and smoothed particle hydrodynamics (SPH) method, are also introduced. Along with the introduction of modelling techniques, relevant design principles and design guidance are also covered. Thus this book can also serve as a handbook for structural engineers. A feature of this work is that it introduces advanced and complicated theory in a more understandable and practical way.

In real design practice, we analyse the structure with an advanced program to gain a level of confidence, such as a ball-park figure for the size of the structural members, but when we start the design we will still follow a code of practice, even though some are quite conservative. Advanced modelling is particularly complementary to current design guidance in those areas where it is still not clear. Therefore, this book will help readers understand the balance between analysis and design.

1.2   Main structural design problems

As a structural engineer, one is required to design different type of buildings such as tall buildings, bridges and space structures. Each type of structure features different structural design problems that a structural engineer needs to pay special attention to during their design. This book covers almost all the important design issues in modern construction projects. In this section, a brief introduction to these different structural problems will be given.

In tall building design, the main issue is the design of the lateral stability systems. In Chapter 3, a detailed introduction to the different lateral stability systems – such as out-triggers, tubular systems and bracing systems – will be given in addition to information on how to model them effectively.

Earthquake design is important for buildings being built in high seismic activity areas. This is covered in Chapter 4. The major earthquake analysis methods – such as response spectrum analysis, time history analysis and push-over analysis – are introduced and modelling examples in SAP2000 are also provided.

Progressive collapse has become another important issue since 911: Chapter 5 covers this topic. The design methods provided by the design guidance are introduced. Different analysis procedures, such as linear static, nonlinear static, linear dynamic and nonlinear dynamic analysis, are explained. A modelling example of nonlinear dynamic progressive collapse analysis is demonstrated using SAP2000.

Aside from conventional loading, blast and fire are other possible threats to the building and its occupants, and Chapters 6 and 7 cover these issues. How to represent these types of special loading and the corresponding design guidance are introduced. In Chapter 6 a new technique in modelling blast or impact effect, the SPH method, is introduced and a modelling example of SPH analysis using Abaqus® is demonstrated. In Chapter 7, a modelling example of heat transfer analysis of a structure is demonstrated.

For space structures, the main design issue is member buckling and overall buckling of the structure; the analysis theories underpinning buckling analysis are introduced in detail in Chapter 8 and corresponding modelling examples are also given.

This chapter also covers a special topic on Tensegrity domes, which have a different structural form to conventional long-span space structures.

Regarding bridge structures, different structure types – such as the beam bridge, cantilever bridge, suspension bridge and cable-stayed bridge – are introduced in Chapter 9. One of the main design issues for bridges is designing the structure under moving load from vehicles, and the corresponding design guidance is introduced. Modelling examples of two famous bridges, Millau Viaduct and the Forth Bridge, are also given.

Foot-induced vibration is a critical issue for the design of foot bridges and hospitals. This is because foot bridges are prone to vibration problems, and hospital buildings have strict requirements for vibration prevention. The vibration problem and corresponding modelling examples are covered in Chapter 10.

1.3   Introduction of finite element method

Numerical methods are fundamental to most analysis software. There are extensive numerical methods that have been developed so far, which include the finite element method, boundary element method, finite difference method, finite volume method and the meshless method (such as the SPH method).

In structural analysis, the finite element method (FEM) is one widely used numerical method. Therefore, it is important for a structural engineer to have some basic knowledge of FEM. In this section, the basic principles of the finite element method will be introduced. Another numerical method, the SPH method, which is used for the analysis of blast or impact loading, will be introduced in Chapter 6.

1.3.1   Finite element methods

The development of the finite element method can be traced back to Courant (1943) in his investigation of the torsion problem. The term 'finite element' was first coined by Clough (1960) and research on this topic has also been conducted by other researchers such as Turner (1956). This numerical method was first used in structural analysis problems in civil and aeronautical engineering. Following that, FEM was applied to a wide range of engineering problems, and most commercial FEM software packages – such as Abaqus®, ADINA and ANSYS – were developed in the 1970s.

FEM is one of the numerical techniques for finding approximate solutions for differential equations with different boundary conditions. It divides a structure into several small elements, named finite elements, then reconnects these elements at their nodes through the compatibility relationships between each element, as the adjacent elements share the same degree of freedom (DOF) at connecting nodes (as is shown in Figure 1.1). The methods for connecting these simple element equations are provided to approximate a more complex equation over a larger domain. The displacement of each node can be determined by a set of simultaneous algebraic equations. Through the compatibility relationship, the displacement can be interpolated over the entire structure.

The major steps of a finite element model can be identified as follows:

1. Select element types.
2. Discretise the structure into pieces (elements with nodes).
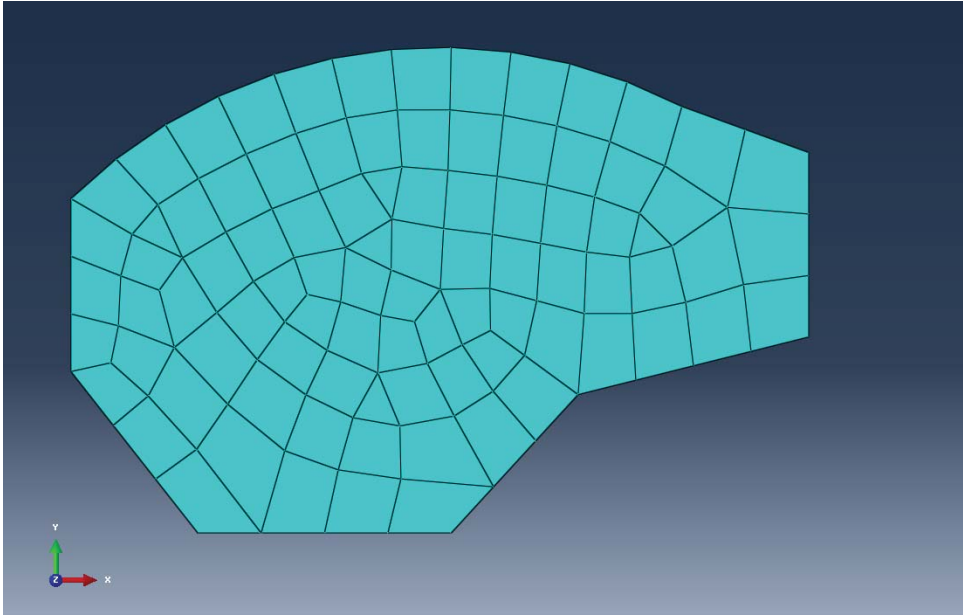
Fig. 1.1 **Finite element mesh in Abaqus®.**
Abaqus® screenshot reprinted with permission from Dassault Systèmes. Abaqus® is a registered trademark of Dassault Systèmes and/or its subsidiaries.

3. Assemble the elements at the nodes to form a set of simultaneous equations.
4. Solve the equations, obtaining unknown variables (such as displacement) at the nodes.
5. Calculate the desired quantities at elements (strains, stresses etc.).

1.3.2 Finite element types

Based on actual engineering problems, there are different types of finite elements available that can be used in the analysis. The key difference between these different types of elements is in their degrees of freedom, and hence a suitable choice requires a reasonable assumption of structural behaviour by the engineer.

(a) The truss element, as shown in Figure 1.2, is assumed to only resist axial force, not bending load and shear load; it is also called a two-force member, as it only has two internal forces at each end. It is usually used for modelling trusses (either bridge trusses or roof trusses) and space structures (domes, vaults etc.). Figure 1.3 illustrates an example of a roof truss model.
(b) The frame element is shown in Figure 1.4; it can model axial, bending and torsional behaviour and is usually used to model beams and columns in a multi-storey building. In most finite elements programs there is also a beam element available, and the axial force is ignored in this type of element. In most structures, such as multi-storey buildings, the axial force in the structural beams
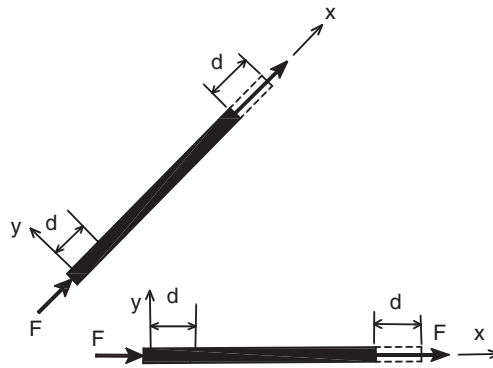
Fig. 1.2   Truss element.
Drawn in AutoCAD$^{\circledR}$. Autodesk with the permission of Autodesk, Inc.
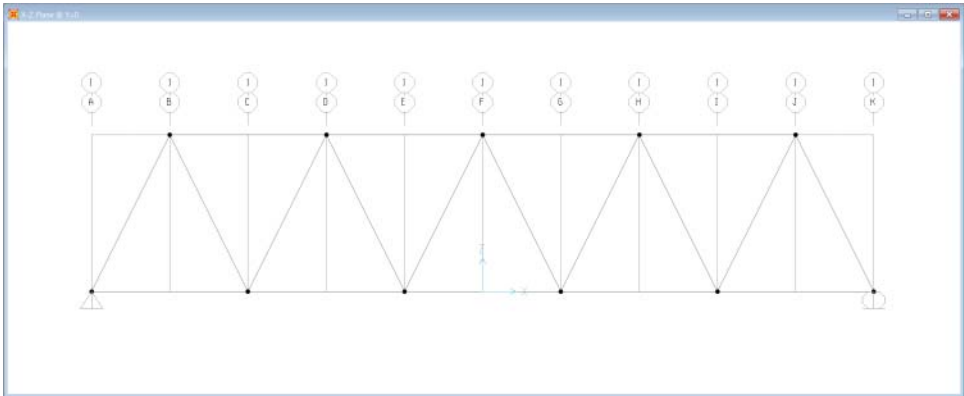
Fig. 1.3   A roof truss model in SAP2000.
SAP2000 screenshot reprinted with permission of CSI.
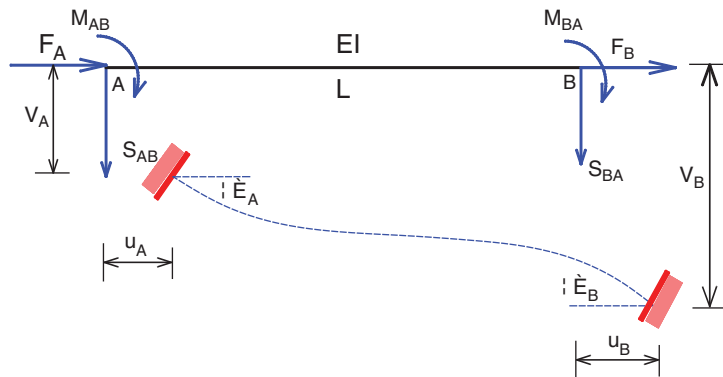
Fig. 1.4   Frame element.
Drawn in AutoCAD.
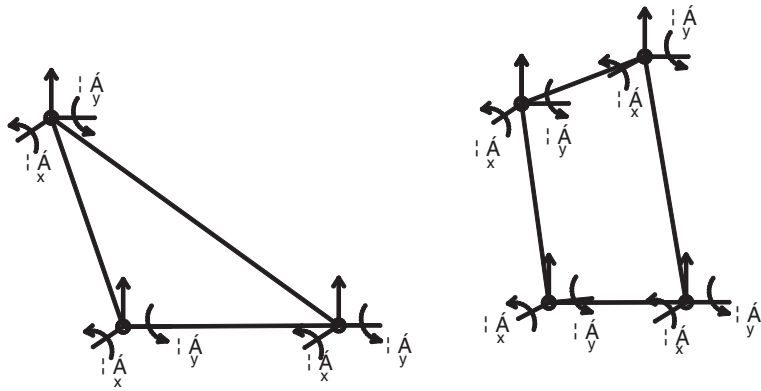
Fig. 1.5   **Typical plate element.**
Drawn in AutoCAD.

can be ignored; therefore the beam element is accurate enough to model this type of structural element. For columns, as they withstand high axial loads and are also subject to bending and shear, frame elements are ideal for the simulation of this type of structural element.

(c) The plate element is used to model flat structures whose deformation can be assumed to be predominantly flexural. Plate elements only consider out-of-plane forces; this means in-plane stress, such as the membrane effect, is not considered. Typical plate elements are shown in Figure 1.5.

(d) The shell element is used to model both in-plane and out-of-plane forces. It consists of two types of conventional shell elements: 2D shell elements and 3D continuum shell elements.

The nodes of a conventional 2D shell element, however, do not define the shell thickness; the thickness is defined through section properties.

3D continuum shell elements resemble 3D solid elements (this will be introduced later) in that they discretise an entire 3D body yet are formulated such that their kinematic and constitutive behaviour is similar to conventional shell elements.

Although 3D continuum shell elements are more accurate in terms of modelling, in most engineering problems conventional 2D shell elements provide sufficient accuracy. In structural analysis, in most cases, 2D shell is effective for analysing structural members such as floor slabs or concrete shell roofs.

Figure 1.6 illustrates some typical shell elements. Figure 1.7 shows an example of a typical floor modelled with 2D shell elements.

(e) The 3D continuum element (solid element), as shown in Figure 1.8, can be used to model fully 3D structures such as dams and steel connections. Solid elements are the most accurate way to represent a real structure, however their computational cost is very high. Depending on the dimension of the structure and the engineering problems you want investigate, solid elements are not suitable for modelling a space structure or multi-storey building due to the high computational cost, it is better to model the full behaviours of a structural element, such as a composite connection (as shown in Figure 1.9). There are some structures for which 3D stress analysis is critical, one example is dams. As thermal stress, shrinkage and
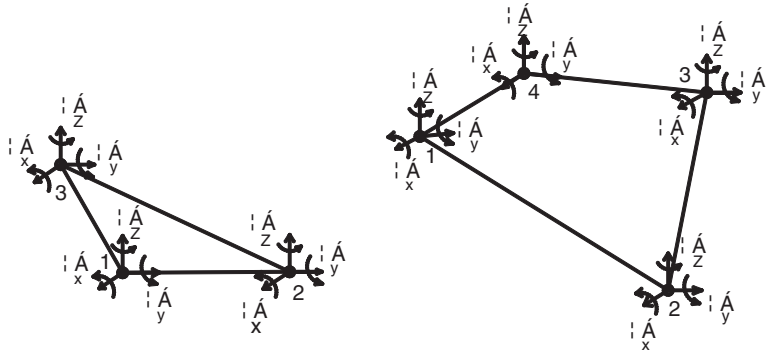
Fig. 1.6   Typical shell elements.
Drawn in AutoCAD.
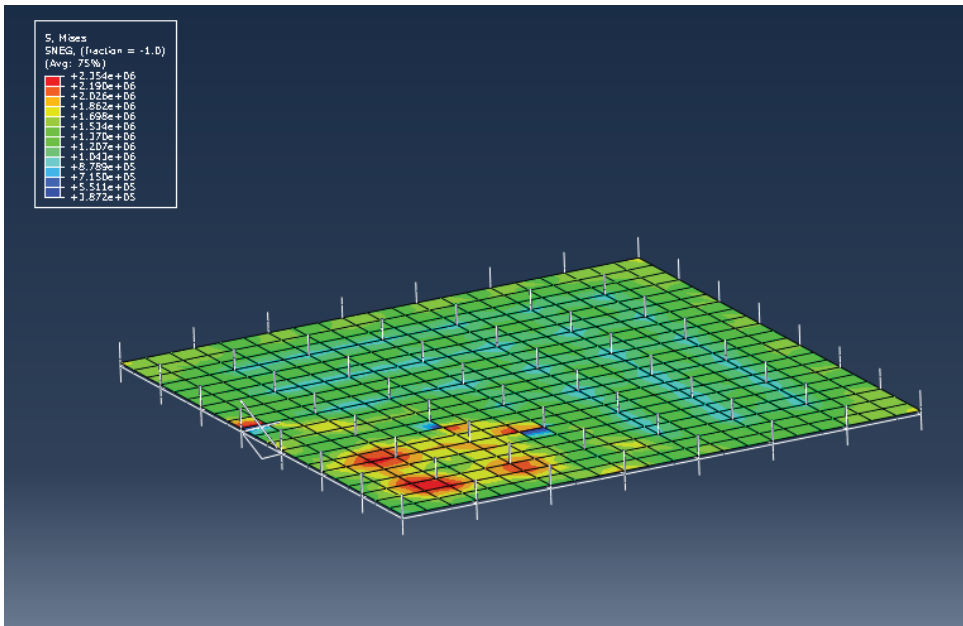
Fig. 1.7   A typical floor modelled with 2D shell elements in Abaqus®.
Abaqus® screenshot reprinted with permission from Dassault Systèmes. Abaqus® is a registered trademark of Dassault Systèmes and/or its subsidiaries.
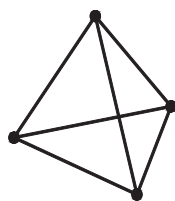
Tetrahedral elements   Hexahedral elements
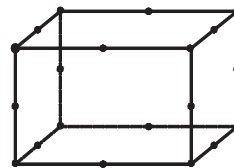
Fig. 1.8   Typical 3D solid elements.
Drawn in AutoCAD.
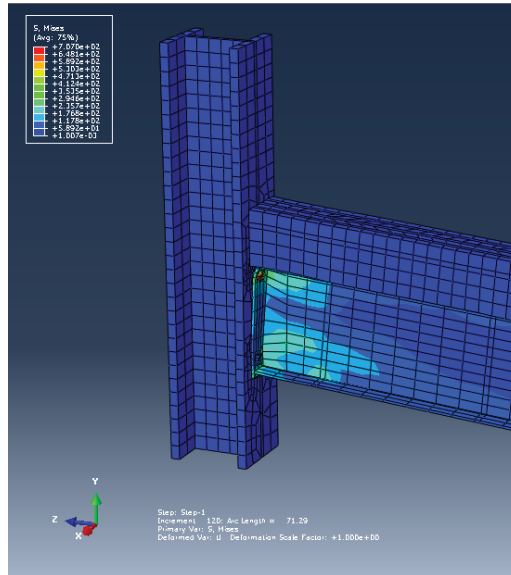
Fig. 1.9   A composite connection modelled using 3D continuum elements in Abaqus®. Abaqus® screenshot reprinted with permission from Dassault Systèmes.

creep are critical in the design of a dam, only 3D stress analysis can accurately represent its 3D behaviour.

There are enormous numbers of 3D solid element types in analysis packages, for example Abaqus® has over 100 types of 3D elements such as C3D10 and C3D20; one of the main differences among them are the integration node numbers for each element. An element with the same shape can have different integration nodes. Figure 1.8 shows some typical 3D solid elements.

1.4   Conclusion

In this chapter, we have discussed typical engineering design problems for different types of structures. We have also introduced the finite element method and its application in structural analysis.

References

Clough R.W, 1960. "The finite element method in plane stress analysis", *Proceedings of American Society of Civil Engineers, 2nd Conference on Electronic Computations*, Vol. 23, American Society of Civil Engineers, pp. 345–378.

Courant R., 1943. "Variational methods for the solution of the problems of equilibrium and vibrations", *Bulletin of the American Mathematical Society*, 49: 1–23.

Turner M.J., Clough R.W., Martin H.C., and Topp L.C. 1956. "Stiffness and deflection analysis of complex structures", *Journal of Aeronautical Science*, 23: 805–823, 854.

2 Major modelling programs and building information modelling (BIM)

2.1 Fundamentals of analysis programs

In this section, how to effectively select a suitable analysis package and basic analysis procedures will be introduced.

2.1.1 Selection of correct analysis packages

There are many modelling programs, such as ANSYS, Abaqus®, ETABS and SAP2000, available to the construction industry. The key to choosing a suitable analysis program in design practice is based on the feature of the specific engineering problems, whether it can provide a suitable model to replicate the structure and its ease of use. This is particularly important as the complexity of the analysis increases; certain software packages may be more suited to a particular application.

All analysis packages used in the design practice can be divided into two main categories: general purpose programs and design orientated programs.

General purpose program such as Abaqus® and ANSYS are suitable for more advanced analyses. For special engineering problems, such as blast and fire, general purpose programs are the right option as they have extensive material models, elements and different solvers (such as explicit or implicit solvers). This has resulted in many consultant companies starting to use these types of software in their design practice in recent years. The disadvantage of this type of package is that design checks against code are not always available.

For a conventional structure problem, design-orientated software packages such as SAP2000, Staad.Pro, Robot and ETABS will be the best option. However, they usually have limited capability to model certain types of structures and have limited types of finite elements compared to general purpose programs. Their advantage is their capacity to carry out design checks based on Eurocodes or any other design guidance that has been already pre-programmed. They can therefore quickly post-process the analysis results and design the structure according to codes of practice. This is one of the biggest differences to general purpose programs. The analysis process is simple and straightforward. For example, a multi-storey building can be modelled using slabs, columns and beams, with loading and supports. After analysis, one can obtain the bending moments or axial force of each structural member and the deflection of the structural members and the whole building. A design check to find the correct size of the structural members of chosen elements is also available. These programs tend

to use a reduced number of elements and only those suited to a particular type of structural element, say a slab or column; therefore the computational time is greatly reduced.

2.1.2   Basic analysis procedures

The process of modelling and analysis in all of the programs requires engineers to understand the relevant structural behaviour and make correct engineering judgements on the feasibility of the results. An engineer should be able to make judgements on whether the analysis results are correct based on their engineering knowledge.

All the software packages (general purpose and design orientated) have similar principles of operation at their core, which can be summarised into three stages:

1. PRE-PROCESSING is the creation of a model of the proposed structure. The geometry, types of finite elements and material properties are assigned at this stage.
2. PROCESSING is the job analyzed by the solver. Here, the unknown values, such as the displacements and stress, can be solved. Depending on the type of problem, different types of analysis procedures and solvers (such as implicit or explicit) can be used.
3. POST-PROCESSING is the final stage where interpretation of the result can be done, for example strain and stress, bending moments, axial forces, deflections, modal shapes and so on can be checked .

2.2   Building information modelling (BIM)

Building information modelling (BIM) has attracted a lot of interest in recent years in the construction industry. The U.K. government is currently encouraging all the disciplines within the construction industry to use it as a standard model for building projects. The purpose of BIM is to set up models using a program such as Revit, and share these models between all professionals in the design team, including architects, structural, electrical and mechanical engineers. Using programs such as Revit, updates of drawings can be automatically reflected to each discipline, thereby enabling the continued management of information on the building's components. During the design phase and the lifecycle of the building, the architectural, structural and MEP engineer (mechanical, electrical and plumbing) drawings are generated through a 3D draughting program such as Autodesk Revit or AutoCAD.

Figure 2.1 shows a flowchart of a design process using BIM. It illustrates BIM is not a single software tool, but a whole package including the draughting program mentioned above and a structural analysis program such as Abaqus® or SAP2000. The BIM process facilitates design, visualisation, simulation, analysis and collaboration to be combined into a single model that can provide a rich source of information about the project.

It can be seen that BIM facilitates a new way of working by creating designs as all the information is automatically built into the model. All the different stakeholders have access to the central model made in Revit; therefore, regardless of how many times the design changes, the data remains consistent and coordinated. It links together all
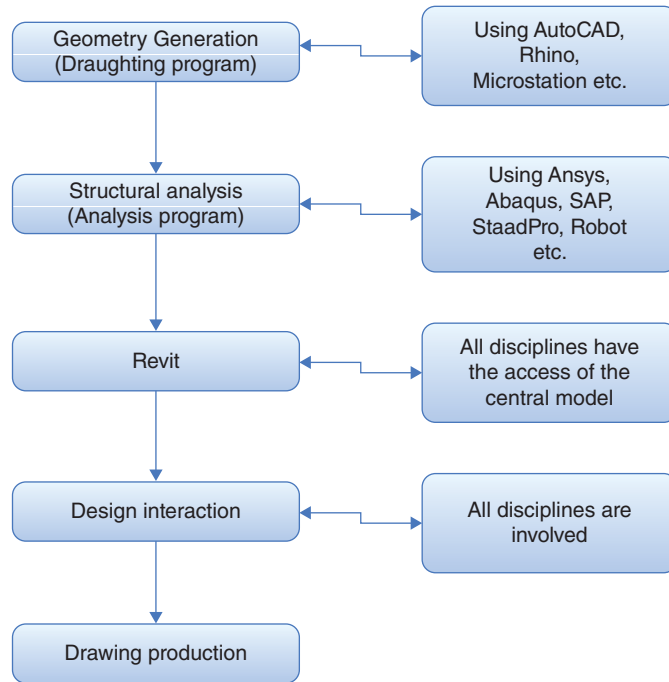
Fig. 2.1   A flowchart of the design process using BIM.

the different parts, through the lifecycle of the process, from design to construction, into one workflow. Therefore, all stakeholders can gain a much greater understanding. Using BIM, the designs can also provide analysis on how to improve every aspect of the project, such as material costs saving.

2.3   Main analysis programs in current design practice

In this section, the most popular software used in current design practice, such as ETABS, SAP2000, Abaqus® and ANSYS, will be introduced. This is to give readers some general background information.

2.3.1   Abaqus®

Abaqus® (2014) is a registered trademark of Dassault Systèmes and/or its subsidiaries. It is a general purpose finite element program originally used in the aerospace and automotive industries. However, over the past decades, it has become a popular program in the civil engineering industry. It uses the open-source scripting language Python for scripting and customisation. Abaqus®/CAE uses the fox-toolkit for GUI (graphical user interface) development.

It consists of three main products:

• Abaqus®/CAE is a pre-processing and post-processing software used for both the modelling and analysis of models and visualises the analysis result.

- Abaqus®/Standard is a finite-element solver that employs a traditional implicit integration method, therefore it is ideal for solving static problems.
- Abaqus®/Explicit is a finite-element solver that employs an explicit integration method to solve highly nonlinear systems with complex contacts under transient loads, therefore it is ideal for solving dynamic problems such as earthquake, blast and so on.

Its use in our civil engineering area is limited, as the civil engineering specific function is not available unless there is further development using GUI or API (application programming interface).

2.3.2 ANSYS

ANSYS is a general purpose finite element program that is used in almost all engineering disciplines. For structural engineering, ANSYS supports a wide range of material models, and enables the application of static as well as dynamic loads. It can perform both linear and nonlinear analysis with respect to material nonlinearity (i.e. plasticity) and geometric nonlinearity (e.g. cable structures).

However, similar to Abaqus®, its use in civil engineering is limited unless there is further development using GUI or API (application programming interface). It is suitable for analysis of complicated structural problems.

2.3.3 SAP2000

SAP2000 is civil engineering software designed by Computer and Structures Inc. (2011). It is used for designing and analysing different types of structural systems such as bridges and buildings. It features built-in modelling templates, design optimisation procedures, advanced analysis options and customisable output reports.

In the experience of the author, it is especially powerful for modelling 3D complex structures such as space structures due to its 3D object based graphical modelling environment and a variety of analysis and design options. Complex models can be generated and meshed with built-in templates. Integrated design code features can automatically generate wind, wave, bridge and seismic loads. It also features automatic steel and concrete design code checks per US, Canadian and international design standards.

It can perform different kinds of analyses such as Eigen and Ritz analyses, catenary cable analysis, buckling analysis, progressive collapse analysis, nonlinear and time history analyses and so on. It can also simulate velocity dependent dampers, base isolators, support plasticity and nonlinear segmental construction analysis.

2.3.4 ETABS

ETABS is civil engineering software designed by Computer and Structures Inc. (2008, 2011). It has been designed especially for multi-storey building analysis. It is storey-based software, which means you can define a typical floor and duplicate it as many storey levels as possible. Therefore, it is extremely powerful for modelling tall buildings. The CAD drawings of floor layout can be imported into ETABS and be

used as templates onto which ETABS objects may be overlaid. In addition, a 3D CAD drawing can be converted directly into ETABS as well.

It supports nonlinear modelling, construction sequencing modelling and time effects (e.g. creep and shrinkage) modelling. Similar to SAP2000, a design check can be performed on various types of structural forms, such as steel and concrete frames (with automated optimisation), composite beams, composite columns and so on on. It also provides comprehensive report output and schematic construction drawings of framing plans, schedules, details and cross-sections.

2.3.5   *Autodesk robot structural analysis professional*

This is a product from Autodesk (2014). It is part of the BIM package provided by Autodesk. It can be linked to Revit (a draughting program that will be introduced later) The link supports the iterative design process and the exchange of data.

It provides structural engineers with advanced building simulation and structural element analysis capabilities for large, complex structures. It has capacities such as flexible solving options, wind load simulation and advanced auto-meshing. Similar to other design-orientated programs, it can perform reinforced concrete design and steel design modules based on more than 40 international codes. It uses Microsoft Component Object Model (COM) technologies, enabling an open and flexible API (application programming interface). From the author's own experience, one of the special features of Robot is the auto-mesh, and it has a strong capacity to mesh the slab and core walls.

2.3.6   *STAAD.Pro*

STAAD.Pro is a structural analysis and design computer program provided by Bentley Systems (2014). It supports several steel, concrete and timber design codes. It can make use of various forms of analysis from the traditional 1st order static analysis, to 2nd order p-delta analysis, geometric nonlinear analysis or buckling analysis. It can perform dynamic analysis such as modal analysis, time history and response spectrum analysis.

In recent years it has become part of integrated structural analysis and design solutions mainly using an exposed API called OpenSTAAD to access and drive the program using a VB macro system, which is a programmable macro system. In addition, it has direct links to applications such as RAM Connection and STAAD.Foundation.

2.4   Major draughting program

To be able to effectively model complex structures, it has become increasingly important in current design practice for a civil engineer to be able to use some of the widely used draughting programs. In most cases, it is easier to set up the model in these draughting programs and import it into the analysis program, rather than set up the model directly from the analysis program.

In this section, popular draughting programs such as AutoCAD, Bentley, Rhino and Revit will be introduced. It worth knowing that, for programs such as AutoCAD and Rhino, a new feature of parametric modelling has been introduced, which enables the

engineer to change the dimension and geometry of the model through modifying the value of certain variables rather than redrawing the model. It has therefore become a powerful tool in modern design practice.

2.4.1 AutoCAD

AutoCAD is a 3D computer-aided design (CAD) and draughting software developed by Autodesk, Inc. (2014). In the civil engineering industry, AutoCAD is used across a wide range of professionals, including architects, project managers and structural engineers and mechanical engineers. It is heavily used by architects and draught technicians.

One of the new features in the latest AutoCAD version is the parametric modelling method, which means one can change the geometry of the model by simply changing the correspondent variables rather than redrawing the model. Readers can find the parametric module easily from the main menu of AutoCAD.

2.4.2 Autodesk Revit

As introduced earlier, Autodesk Revit is part of the Building Information Modelling software developed by Autodesk (2014). It can be used by architects, structural engineers, MEP engineers, designers and contractors. It allows users to design a building, structure and its components in 3D, annotate the model with 2D draughting elements and access building information from the building models database. Revit also developed 4D BIM with tools to plan and track various stages in the building's lifecycle.

As a student or an educator, you are allowed to download both AutoCAD and Revit from this link http://www.autodesk.com/education/free-software.

2.4.3 Rhino3D

Rhinoceros (Rhino) is a commercial 3D modelling software developed by Robert McNeel & Associates (2014). The software is commonly used for industrial design. It is currently widely used by architectural design companies and some structural design consultants. It specialises in free-form nonuniform rational B-spline (NURBS) modelling. Therefore, it is easier to handle complex curvature in the building geometry.

The major feature of Rhino is its parametric modelling technique, as it features plug-in development. An engineer can further extend the function of the program by developing the plug-ins. The Grasshopper plug-in for computational design is an important feature. Parametric modelling is made available through Grasshopper's platform, which means one can change the dimension and geometry of the model by simply changing the corresponding variables in the programming rather than redrawing the model whenever it requires a change. This is extremely useful at the stage of scheme design, when the design is constantly changing according to the requirements of the architects and clients. Many architects use parametric modelling tools and some structural engineers are starting to use this new technology as well.

Rhino also features a scripting language, based on the Visual Basic language, and an SDK that allows reading and writing of Rhino files directly.

2.4.4 Bentley MicroStation

MicroStation is one of the major CAD software products for design and draughting developed by Bentley Systems (2014). MicroStation is a platform for architectural and engineering software packages. It generates 2D/3D vector graphic objects and elements. Bentley was probably one of the first companies in the world to develop CAD software.

2.5 Method to model complex geometry

Although there are pre-processing modules available for most of the analysis packages, many of them have limit capacities. Therefore, in reality, for most complicated structures, it is difficult to create the geometry of the structure and set up the model directly from the analysis software we have just introduced. In addition, most of the geometry originally comes from architecture models and drawings; therefore, importing a geometrical representation (or idealisation – it rarely needs to be an *exact* replica of geometry) of the required structure from a draughting program such as AutoCAD or Rhino is an effective tool, as most architects will provide the structural engineers with a 3D model or 2D drawings created using AutoCAD, Revit, Rhino and so on.

The steps to set up a model using a draughting program are as follows:

1. First, a reasonable simplification should be made to the architectural model and drawings. The points, lines, surfaces and volumes should be modified. Some unnecessary elements, such as fittings and partitions, should be deleted and only structural elements kept. In most cases, only points and polylines are required, as the size of structural members such as slab thickness and beam size can be defined in the analysis software quite straightforwardly. A necessary explode of the entities is required in certain cases, to make sure the analysis program can recognize them after importing.

    Remember, the draughting programs are only to help you get the geometry of the structure. Most analysis programs cannot recognise complicated elements, such as solid slabs or 3D beams, therefore reasonable simplification is the key to successful modelling.

2. Second, the geometry will be imported into an analysis package. Depending on the features of the analysis program, different techniques are involved in this procedure and the relevant techniques will be introduced here.

3. Finally, the geometry is assigned various attributes, such as material properties, geometric properties of the finite elements (depth of a beam element, thickness of a shell element etc.) supports and constraints and loads in the analysis packages.

Over the past decade, the use of graphical user interfaces (GUIs) has opened the doors to a wider audience by making it easier to develop models through the application of geometry.

In the remaining part of this section, the method of setting up a complex model using draughting programs and details on how to import them into an analysis program such as SAP2000 or Abaqus® will be introduced.

2.5.1 Import geometry into SAP2000

In this section, we are going to introduce how to set up models in a draughting program such as AutoCAD or Rhino and import them into SAP2000.

Import geometry from AutoCAD

Drawing files may be imported from AutoCAD into SAP2000 using the following procedure: an example on how to set up a Tensegrity unit will be illustrated here. A detailed modelling example of a Tensegrity structure will be given in Chapter 8.

1. Create a 3D model within AutoCAD as shown in Figure 2.2. If the model originated from an architecture drawing, make sure to further simplify and explode the model into Polylines, and save it into a DXF file.

   Check the origin of the model, to make sure it is close to (0,0,0), this is important, as you may find the model disappears after importing into SAP2000.

   As the structural model may consist of different types of structural members such as shell and beams, so it is important to make different layers in AutoCAD and move or create different types of elements into different layers before importing into SAP2000.
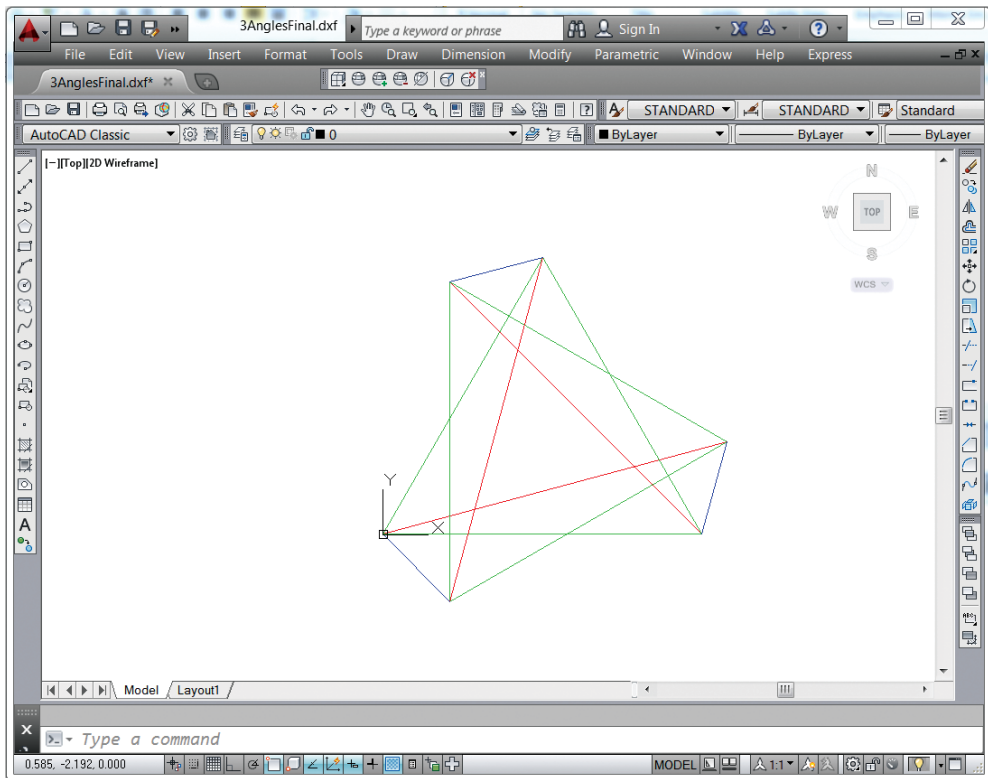
Fig. 2.2   DWG model from AutoCAD.
AutoCAD screenshot with the permission of AutoDesk.
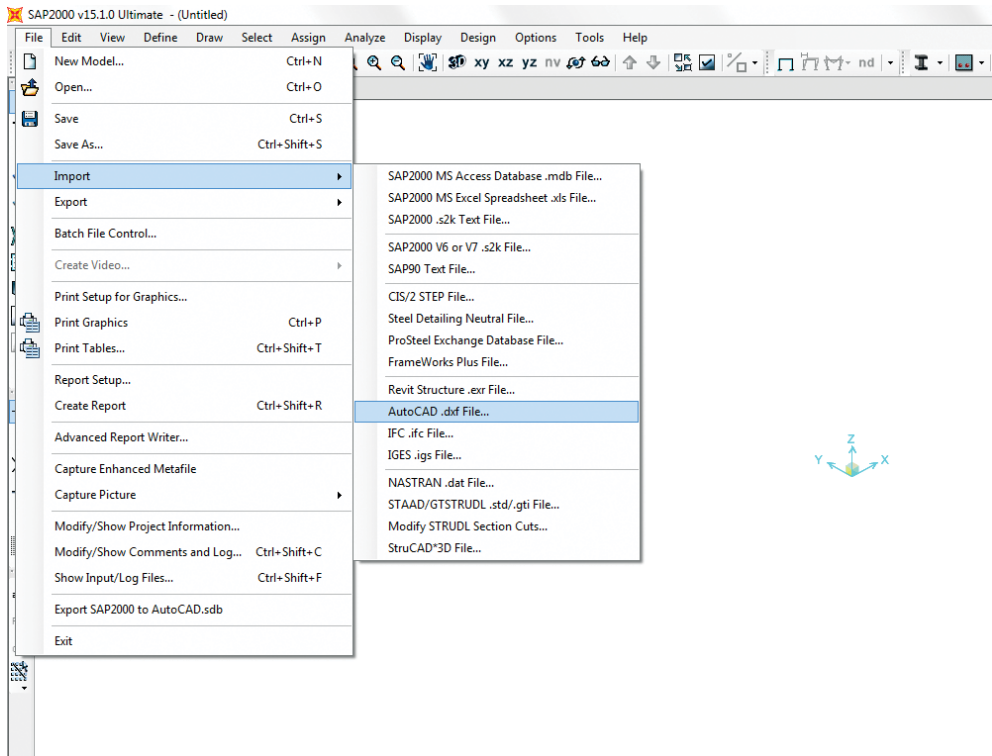
2. In SAP2000, select *File – Import – AutoCAD.dxf File*, as shown in Figure 2.3.
3. Select the DXF Files (*.DXF) from the scroll-down menu, then locate and double-click the DXF file for import into SAP2000.
4. Select the *Global Up Direction* and the units (it is important to choose the correct units at this stage, make sure they are coordinated with your AutoCAD units, otherwise problems will be found after importing; common mistakes that happen here are the model being too large or too small, one can see this in the extrude view after the member size has been defined) , as shown in Figure 2.4:
5. Assign each type of SAP2000 object to a DXF layer (Figure 2.5). Each layer that occurs in the DXF file is listed in the scroll-down menu.
6. Select *OK*, then the DXF file will import as a SAP2000 model, as shown in Figure 2.6.

Set up model with Rhino

As mentioned earlier, Rhino is another effective 3D draughting program that is widely used by architects and engineers. It especially features plug-in development.

Figure 2.7 is an example of how to set up a model in Rhino. Similar to AutoCAD, when setting up or modifying an architectural drawing, make sure to put the different

Fig. 2.4 Import specifications.
SAP2000 screenshot reprinted with permission of CSI.
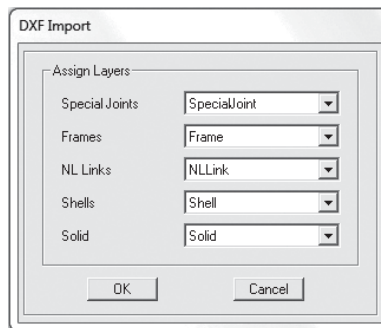
Fig. 2.5 Layer specifications.
SAP2000 screenshot reprinted with permission of CSI.

types of structural elements into different layers. Then save it into a DXF file and follow the same procedure laid out in last Section.

There is also an alternative way to import from Rhino or Grasshopper into SAP2000. Models can be imported into Industry Foundation Classes (IFC), or another intermediate format, and then imported from IFC into SAP2000. Model data may also be selectively imported on a table-by-table basis using interactive database editing, available through *Edit – Interactive Database Editing*. Please refer to the CSI website (2004) for further information.

Set up model in Revit structure

To import models from AutoDesk Revit Structure into SAP2000, a program called CSiXRevit is available to supplement SAP2000 versions V15.1.0/1 and higher. For earlier SAP2000 versions, Revit models may first be imported into Industry Foundation Classes (IFC), or another intermediate format, before being imported from IFC into SAP2000. As an alternative, models may be imported from Revit into ETABS, then saved as a S2K file and imported to SAP2000. Please refer to the CSI website (2004) for further information.
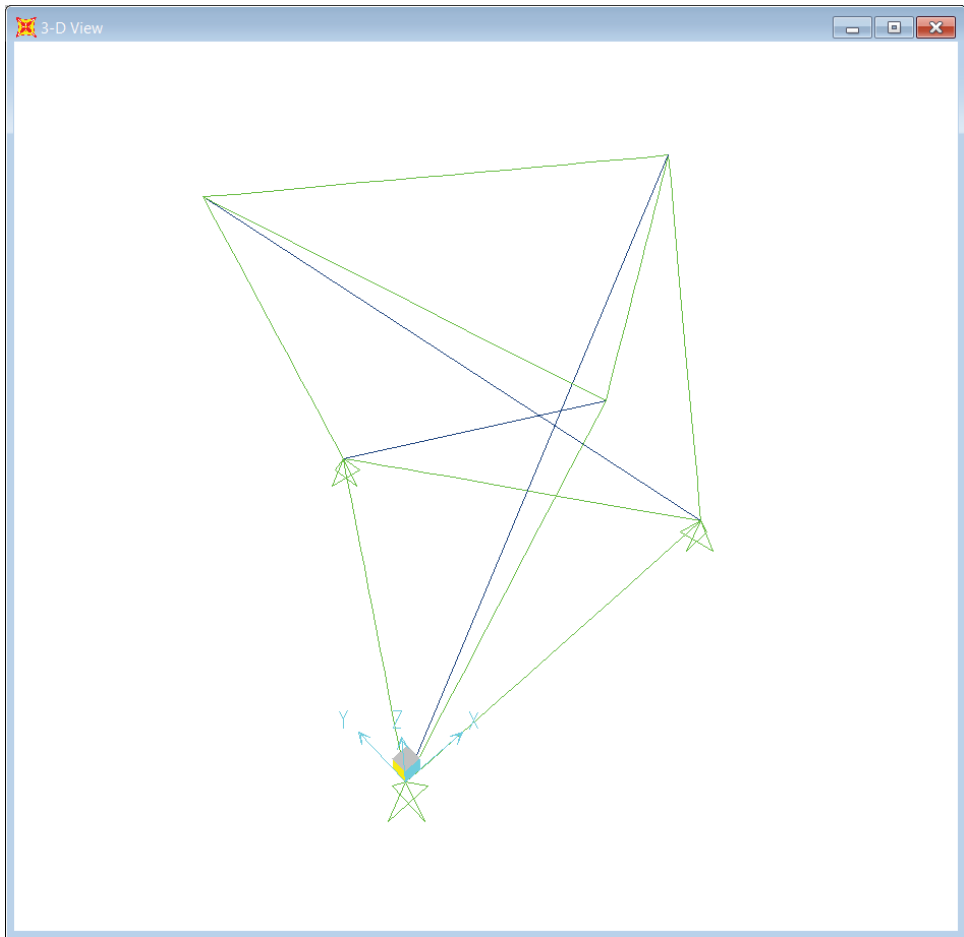
2.5.2   Import geometry into ETABS

As introduced earlier, ETABS is specially designed for modelling multi-storey or high-rise buildings. It is a storey-based software; the engineers need to divide the storeys into certain groups with similar floor geometry. Then, define certain typical floors in ETABS, and duplicate the typical floor throughout the height of the building. How to import the geometry of the floor plan is therefore key in modelling a complicated building.

First, the storey data should be defined. Figure 2.8 illustrates an example of how to define storey data in ETABS.

In real design, the layouts of the floor plate are normally provided by the architect; as a structural engineer, one should simplify the drawing and import the floor plan DXF file into ETABS. Prior to importing, the DXF files should be prepared in AutoCAD as described below:
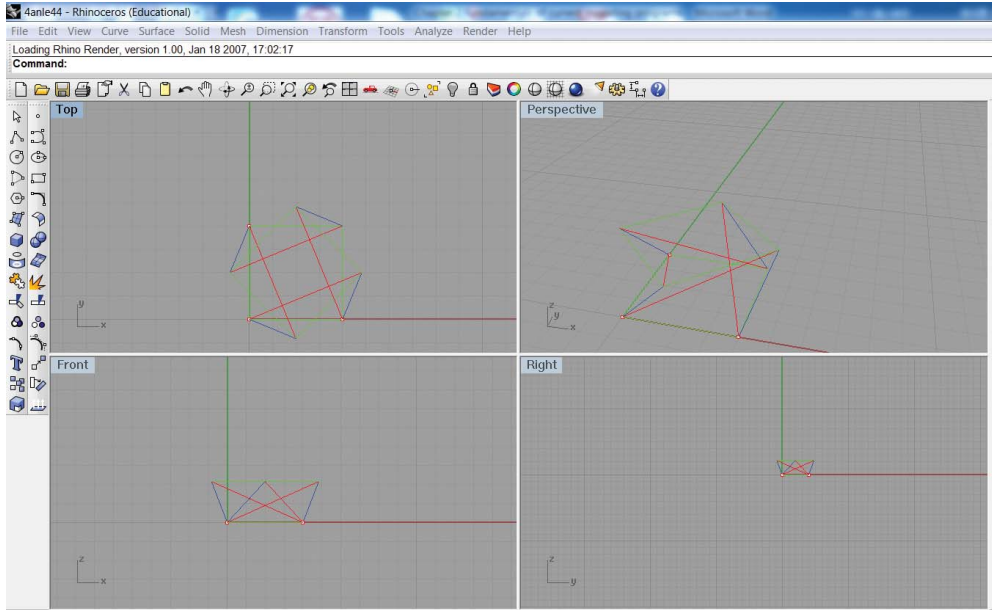
Fig. 2.7   Setting up a 3D model in Rhino.
Rhino 3D screenshot reprinted with permission of McNeel.
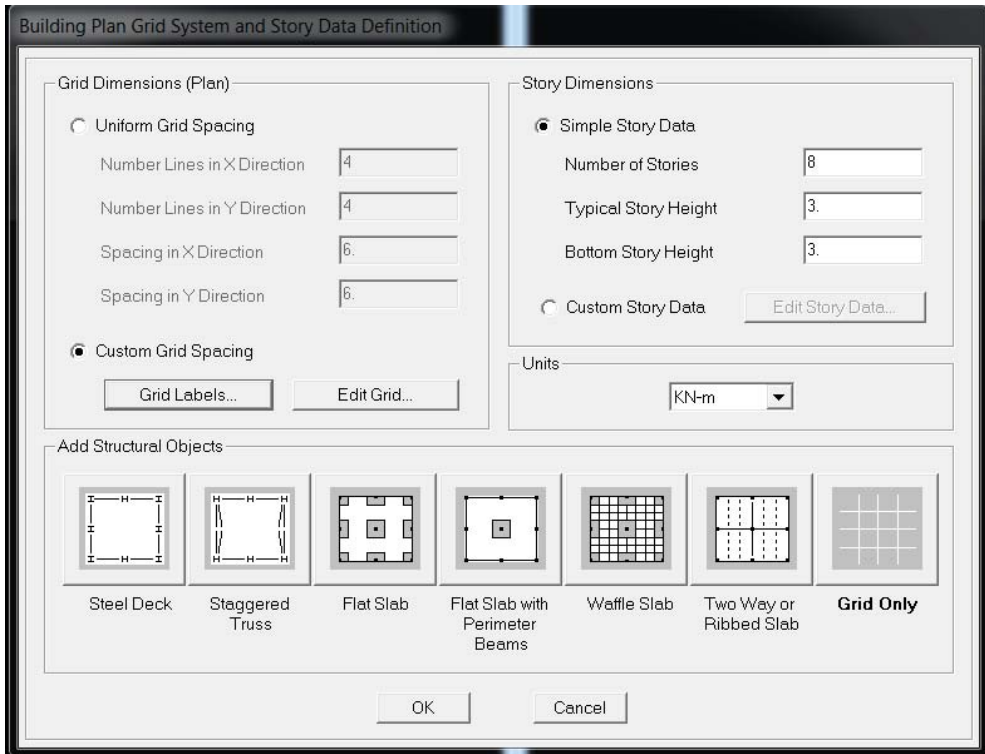
Fig. 2.8   Defining the storey data in ETABS.
SAP2000 screenshot reprinted with permission of CSI.

1. In a new AutoCAD drawing, create different layers for slabs, beams, columns and walls for different floor levels.
2. Attach the relevant architecture drawings as Xref, so you can trace the geometry of the floor plan.
3. Or, directly use the architecture drawings, but delete the unnecessary architectural layers or lines and explode the entities if necessary.
4. Slab edges should be defined as a single closed polyline in CAD on a separate layer.
5. A diagonal line or plus mark should be drawn in CAD to locate the column. This will result in a line element with a centre point located to allow placing of the column in ETABS. This should also be on a separate layer.
6. Centre-lines of shear walls and beams should be defined in separate layers in CAD.
7. Make sure to move the origin of the AutoCAD model at (0,0,0), otherwise one may lose the model following importing into the ETABS.
8. Define the grid line that matches the architectural plans for clarity. ETABS can import grid lines from the DXF plan.
9. Save as a DXF file.
10. Import DXF into ETABS.
11. During the process of importing, select the relevant layers in ETABS.

2.5.3   Import geometry into Abaqus®

Abaqus® has been used in the civil engineering industry for the last two decades. However, as introduced earlier, it is not straightforward to set up a complex structural model directly using the CAE module available in Abaqus®. Fu (2009) has developed a program to convert a complex model such as a tall building from ETABS into Abaqus®.

Here, a detailed introduction will be given according to the different types of entity to be imported.

Imported as 3D frame

3D frames:

1. Create the geometry in AutoCAD or an alternative draughting program such as Rhino. Explode entities if necessary to obtain only polylines and save as a DXF file.
2. Use gCAD3D (a free and open software http://www.gcad3d.org/), import the DXF and save as an IGS file.
3. Import parts as wires with Abaqus®/CAE.

Imported as 3D solids or a 3D shell

Here is an example of how to build a 3D solid entity in an AutoCAD file and then import it into Abaqus®. For 3D solids, a slab has been selected as an example.

1. As shown in Figure 2.9, create rectangular regions and use the command *Extrude* in AutoCAD to make it into a 3D solid, which can then be exported using the *Export* command, as ACIS (*.sat) files in AutoCAD.
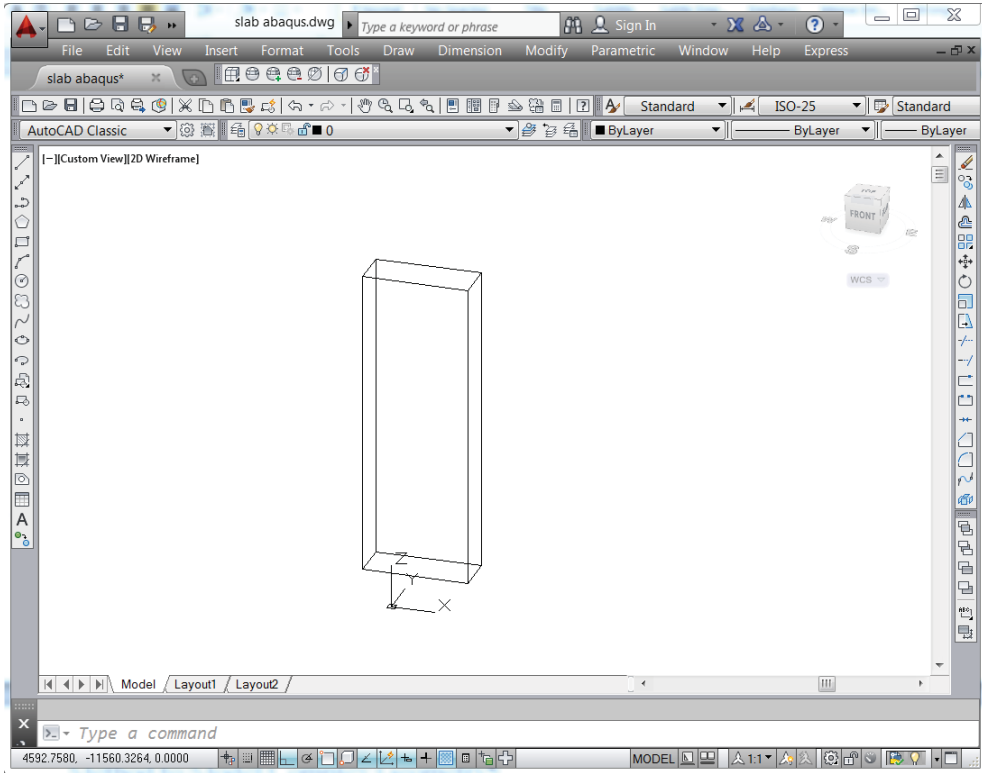
Fig. 2.9   **Setting up a 3D slab in AutoCAD.**
AutoCAD screenshot with the permission of Autodesk.

    2.   Import parts as solids with Abaqus®/CAE.
- In Abaqus®, as it is shown in Figure 2.10, select *File – Import – Part*, then select the **.sat* files generated from AutoCAD.
- This format of files is recognised by Abaqus®/CAE as models that can be imported as 'Part …' or 'Assembly …' in the *Import* command. For this example, the models created in AutoCAD were imported as *Part* into Abaqus®/CAE.
- Name it *slab* and click *OK*, as shown in Figure 2.11.
- A slab is imported into Abaqus® as shown Figure 2.12.

When modelling a part in Abaqus®, you can also define the partitions by drawing them in AutoCAD and importing them into Abaqus®, which provides an alternative way to define the partitions directly in ABAQUS.

Imported files from ETABS into Abaqus®

As mentioned earlier, the pre-processing function of all general purpose programs such as Abaqus® and ANSYS is quite limited. Therefore it is difficult to set up some complicated structures, such as multi-story building models, directly into these general purpose softwares due to the geometric complexity of the structures. To solve this
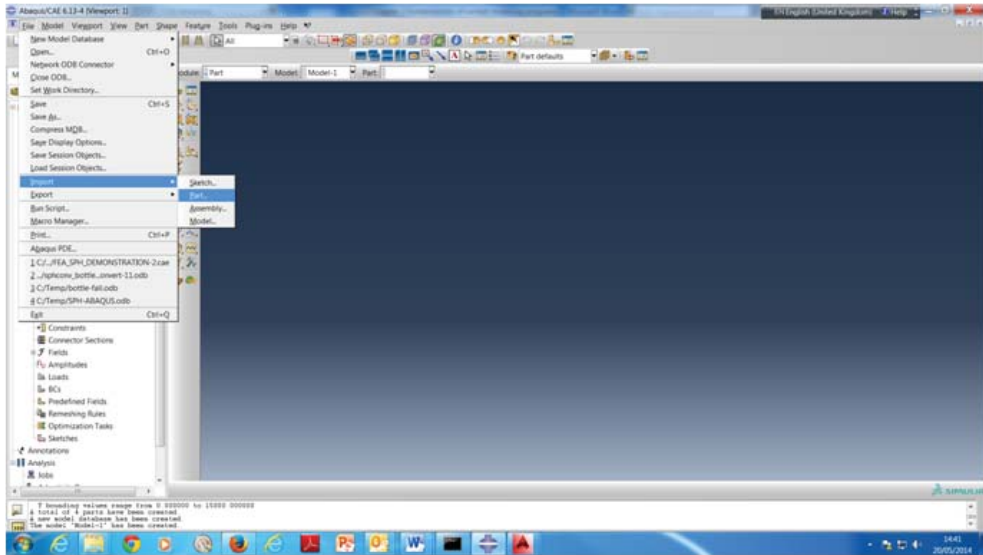
Fig. 2.10   Importing *Part* into Abaqus®.
Abaqus® is a registered trademark of Dassault Systèmes and/or its subsidiaries.
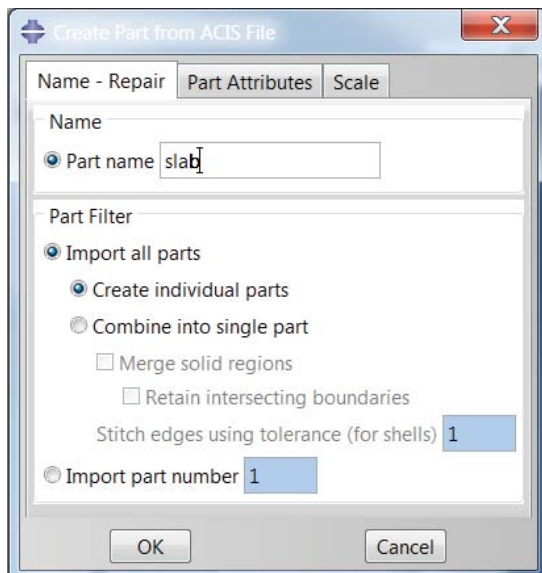
Fig. 2.11   Part importing definition.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

problem, in Fu (2009), ETABS was used to set up the whole model geometrically as it is easier to set up a model and mesh the whole structure with the pre-processing function of ETABS. Therefore, the s2k file of the ETABS model is converted into an Abaqus® input file using the converter program designed by the Fu (2009) with Visual
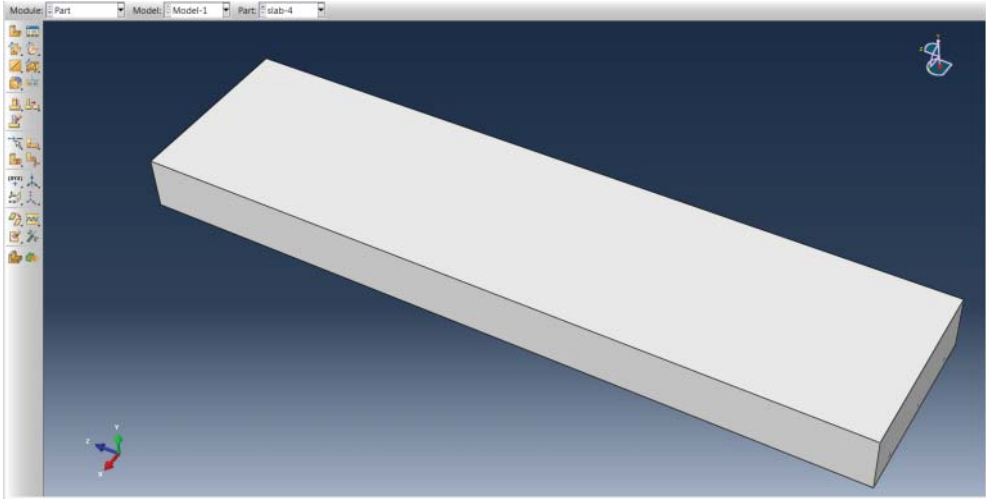
Fig. 2.12 Slab in Abaqus®.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
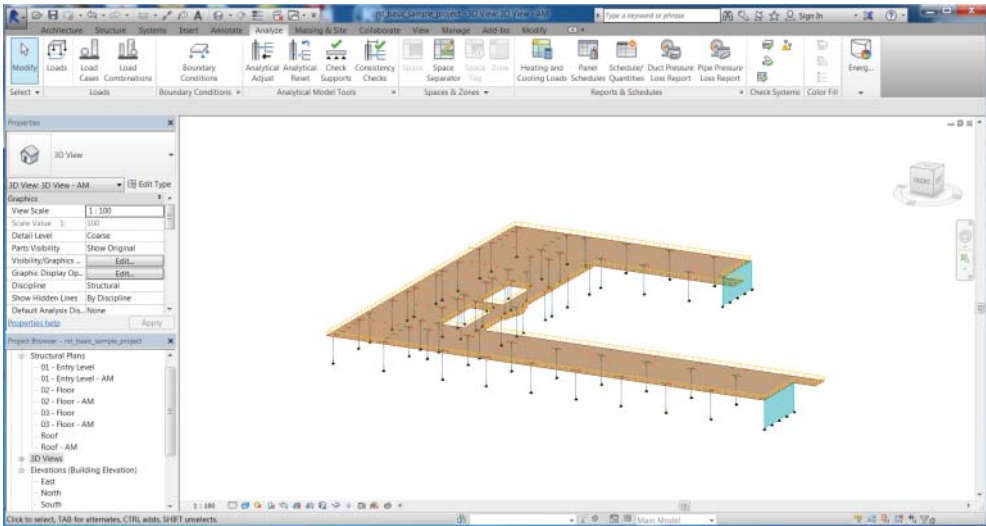
Fig. 2.13 Setting up a model in Revit.
Revit®. Autodesk screenshots reprinted with the permission of Autodesk, Inc.

Basic Language. The converter program can transform exactly all the information in the ETABS model, such as the beam orientation, beam size, shell thickness, shell orientation, moment releases, constrain equation and the mesh of the shell and so on, into the Abaqus® model.
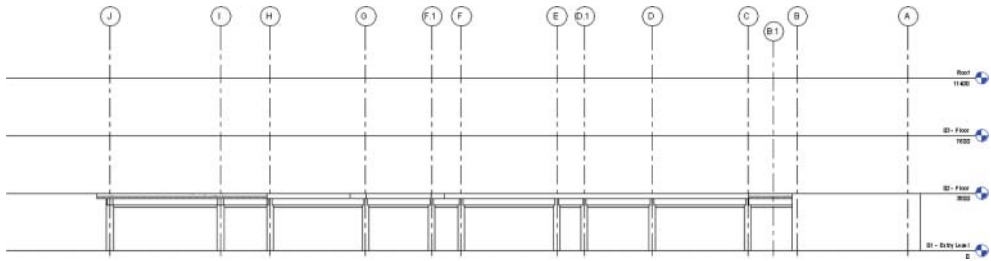
Fig. 2.14　Elevation of the building.
Revit[®].

2.5.4　*Set up model with Revit*

Autodesk Revit is part of the building information modelling software. As shown in Figure 2.13, it consists of architecture, structure, system and analysis modules; the architects and engineers from different disciplines can work on the same model but concentrate in different areas.

When setting up the model, one can also define different floor levels and elevations as shown in Figure 2.14. Therefore, it is quite convenient to model a multi-storey building. The models may be imported from Revit into ETABS and Robot, which makes it an efficient way of modelling.

References

Fu F. 2009. "Progressive collapse analysis of high-rise building with 3-D finite element modeling method", *Journal of Constructional Steel Research*, 65, 6: 1269–1278.

Software and manuals

3D Structural Analysis & Design Engineering Software – STAAD.Pro, Bentley system official website, Bentley System Inc, 2014.

Abaqus[®] Theory Manual. 2013. Version 13.1 Hibbitt Karlsson and Sorensen, Inc., Pawtucket (RI).

AutoCAD, 3D Design, Engineering & Entertaining Software, Autodesk official website, Autodesk Inc. 2014.

Autodesk Robot Structural Analysis Professional, Autodesk official website, Autodesk Inc. 2014.

Bentley System Inc, Bentley system official website, 2014.

Building Design Software, Revit Family, Autodesk official website, Autodesk Inc. 2014.

Computers and Structures, Inc. 2011. Computers and Structures, Inc. official website. Official website summary. Retrieved December 4, 2011.

ETABS Theory Manual. 2008. Version 9.2.0. Copyright Computers and Structures, Inc.

Rhino 4.0 Robert McNeel & Associates (Rhino 5.0) 2014.

3 Tall buildings

3.1 Introduction

The opening of the Burj Khalifa (800 m tall) and the newly proposed 1-km tall building project of the Kingdom Tower in Saudi Arabia have stimulated great interest in design engineers in the design and analysis of tall buildings. In this section, the main design issues of tall buildings, such as gravity and lateral resistance systems, will be introduced, and several modelling examples based on existing buildings such as the Burj Khalifa, the Willis Tower, the Gherkin and Taibei 101 will also be covered.

3.2 Structural systems of tall buildings

The main structural systems of tall buildings consist of gravity load resisting systems and lateral load resisting systems. In real design practice, the main lateral forces that need to be considered are wind and earthquakes. As it has a longer period than low rise buildings, the earthquake load is less onerous. Therefore, in most cases, wind is the governing load in the design.

3.2.1 Gravity load resisting systems

The major components associated with the vertical load resisting systems of high rise buildings are the floor system, columns, transfers beams and shear walls. Shear walls are primarily used for resisting lateral forces but are also used to resist vertical loads.

The floor system is part of the gravity load resisting system that remains relatively similar in both high rise and low rise buildings. However, for tall buildings, it is imperative to minimise the weight of the entire structure. Therefore, lightweight floor systems, such as post-tension slabs, are the dominant selection in the design.

The floor system also plays an important role in resisting lateral loads. Due to its large in-plane stiffness, the lateral loads are transferred through the floor plate via a diaphragm action by allowing a constant pathway to the lateral resisting system, such as core walls and bracings; therefore, the design of adequate floor diaphragms is also an important issue in tall building design. For floor plates with large openings, special attention needs to be paid to these as they weaken the diaphragm action in the floor slab. In some projects, planar trusses need to be employed to enhance diaphragm actions.

3.2.2   *Lateral load resisting systems*

There are various lateral resisting systems currently used in construction practice. Here, a detailed introduction to some typical systems used in tall building design will be given. The features of these lateral resisting systems and their applications in real design practice will be discussed.

Reducing lateral drift is an important design consideration for tall buildings. In general, it is a serviceability issue. Excessive lateral drift, especially inter-storey drift, can damage secondary systems such as partition or curtain walls. Building occupants will also experience discomfort under prolonged cyclical drift (Narim, 1989) and dynamic wind force is the main reason for this. The effectiveness of the lateral resisting system is assessed mainly based on the lateral drift: either the maximum drift at the top of the building or the inter-storey drift. These are the two important factors an engineer needs to check. Commercial programs such as ETABS, SAP2000, Autodesk ROBOT and Staad.Pro all have the ability to check either the overall drift or the inter-storey drift.

In design practice, there is an empirical formula used to check whether the lateral resisting system is sufficient in the scheme design stage, which is

$$\Delta_{max} \leq H/500 \tag{3.1}$$

Where:
Δ_{max} is the maximum lateral deflection
H is the overall height of building.

3.3   Lateral resisting systems and modelling examples

In real construction practice, few tall building use a single type of structural system. In most situations, combinations of structural systems are applied simultaneously in order to overcome the challenges of nature. In this section, different lateral stability systems will be introduced and modelling examples of existing tall buildings utilising some of these lateral resisting systems will be demonstrated using SAP2000.

3.3.1   *Moment resisting frames (MRF)*

The moment resisting frame (MRF) system (as shown in Figure 3.1) is a system that utilises the rigidity of the structure to provide lateral resistance. The beams and columns are rigidly connected, thus it is also called a rigid frame. Both the columns and beams must be designed to be strong when bending. The beam to column connections also need to be specially detailed to accommodate the bending moment caused by the lateral load. Steel and concrete are the most common construction materials for MRFs, both providing flexibility in floor planning.

The disadvantage of the MRF system is that a rigid frame requires expensive moment connections, resulting in an overall expensive building. In addition, it is difficult to control the lateral drift due to the P-Delta effect. The increased sway of the building causes additional bending of the beams and columns. Therefore, the efficient
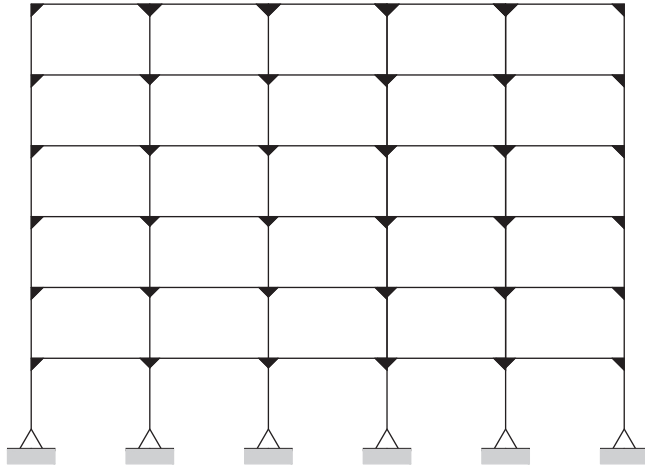
Fig. 3.1 Moment resisting frame.
Drawn in AutoCAD.

design height with no additional lateral load resisting systems required is 30 storeys for steel structures and 20 storeys for concrete structures (Kowalczyk et al., 1995).

In places where seismic activity is higher or there is a greater risk of earthquakes, the MRF should be detailed to make sure that the ductility of the beam to column joints allows for sufficient resistance to the excessive movement and swaying of the structure. The requirement of good ductile properties in design is now seen as a key attribution in resisting excessive movements of a structure.

3.3.2 Shear walls

Shear walls are widely used for both tall buildings and low rise buildings as important structural members in the lateral resisting system. They work as a deep vertical cantilevered beam supported at the ground. They also carry the vertical load together with columns.

In tall buildings, shear walls are generally located towards the centre of the building, normally in the form of a core wall system, as vertical translation systems such as lifts are normally located at the centre of the building. Coupled shear walls, where girders and the floor system join two or more walls together, are also widely used in many projects. This is a very common form of lateral load support system in tall buildings.

It is also quite common for MRFs to be connected with reinforced concrete shear walls or steel trusses, resulting in a shear frame interaction system that provides exceptional lateral rigidity for tall structures, as each system controls the lateral stability of the building at different locations, which provides a more efficient lateral resisting system. These types of systems have the maximum potential to reach a height of 65 floors.

3.3.3 Bracing systems

As shown in Figure 3.2, bracing is one of the most important lateral resisting systems, and it is widely used in different types of low-rise buildings as well as tall buildings.
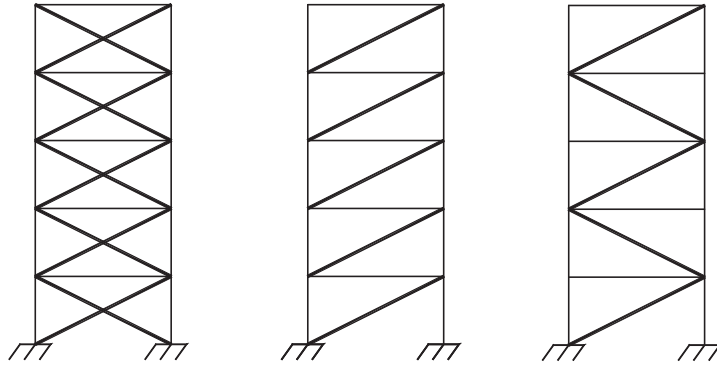
Fig. 3.2 Typical concentric bracings.
Drawn in AutoCAD.

For aesthetic purposes, braced frames are usually situated in the cavity of the walls and the lift-shaft core area. They can be categorised into concentric or eccentric bracing. The differences between them are as follows.

Concentric bracing is oriented such that all members (beams, columns and bracing) meet at a communal point. The brace provides the lateral resistance mainly through axial force in the structural members. Some typical concentric bracing examples can be seen in Figure 3.2.

As shown in Figure 3.3, eccentric bracing comprises both axial loading members (the diagonal members) and bending loading members (the horizontal members). They are heavily used in earthquake zones due to the high ductility they provide.

3.3.4 Outrigger structures

For buildings taller than 70 storeys, the overturning moment that results from the lateral loading increases greatly, therefore purely relying on a single core wall system to resist such loading is not feasible as this would result in massive core walls or much thicker wall thickness. The problem can be solved by using firm horizontal members (such as outriggers) to connect the main core to the exterior columns.
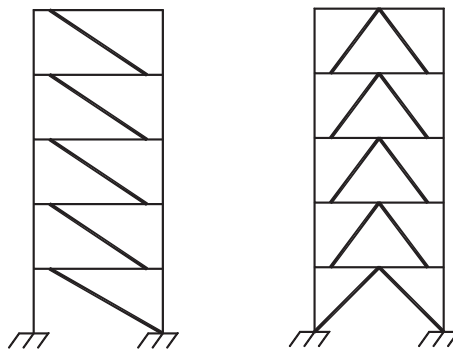
Fig. 3.3 Typical eccentric bracing.
AutoCAD.
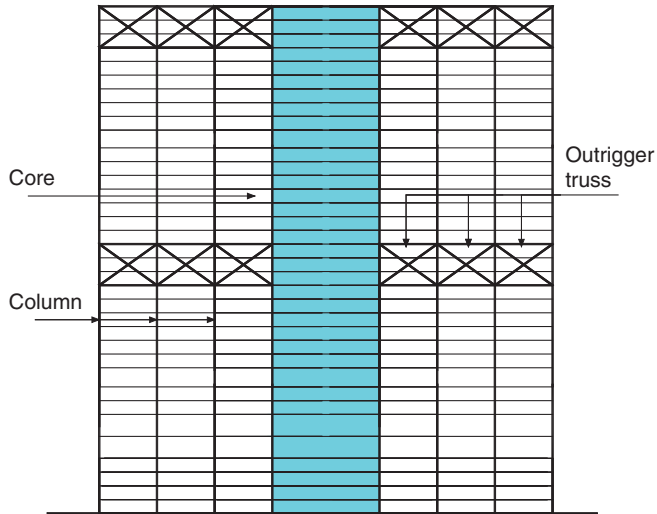
Fig. 3.4   Outrigger structures.
AutoCAD.

The cores are normally located centrally within a building, whereas the outriggers (in steel structures outriggers are generally in the form of trusses) extend out to the outer columns as shown in Figure 3.4.

The outriggers and the outer columns work together as a buttress to support the core wall. This provides a combination of tension and compression within the columns on the outer sides of the structure.

The bending, axial tension and compression of the outer columns connected to the outriggers help resist the external moments of the structure. When lateral loading, such as wind, is acting on the structure, the rotation of the core can also be s controlled due to the restraint from the outriggers. This resistance helps reduce the lateral deflections and moments, which would otherwise cause structural instability of many of the members.

One famous project example of this system is the Shard, London Bridge Tower (as shown in Figure 3.5). It has a core wall at its centre and outrigger trusses at certain levels to connect the central core and outer raking columns.

3.3.5   Tube structures and modelling example of the Willis Towers

Tube structures are one of the major lateral resisting systems in tall buildings. A building is designed to act like a hollow cylinder, cantilevered perpendicular to the ground in order to resist lateral loads (wind, seismic, etc.). A good way to illustrate this system is to imagine a cup that has been put upside down on a desk; one needs a large force to overturn it.

This system was introduced by Fazlur Rahman Khan while at Skidmore, Owings and Merrill's (SOM) Chicago office. A few famous buildings that endorse this design system, are the Willis Towers, the World Trade Centre (collapsed) and the John Hancock Centre.

For this type of system, the exterior framing is designed to be sufficiently strong (normally in terms of rigid beam to column connections) to resist all lateral loads

Fig. 3.5   The Shard.
Photo from Structurae: International Database for Structural Engineering, courtesy of Wilhelm Ernst & Sohn Verlag.

(Ali and Moon, 2007). The perimeter of the exterior consists of closely spaced columns that are tied together with deep spandrel beams through moment connections. The distance between the exterior and the core frames is spanned by beams or trusses. This maximises the effectiveness of the perimeter tube by transferring some of the gravity loads within the structure to it and increases its ability to resist overturning due to lateral loads.

Tubular structures are categorised into several different types, mentioned below.

Framed tubes

Framed tubes are the most widely used tubular system. It is a three-dimensional system that consists of closely spaced columns and deep spandrel beams firmly joined together to make a hollow cylinder. Depending on the structure, the spacing of the columns is quite close, generally 1.5–4.5 m. Spandrel beam depths can range from 0.5–1.2 m.

The effects of shear lag need to be addressed here. Under lateral loading, columns situated near the corners of the tube experience the greatest axial forces whereas the inner columns experience a reduction in axial load and hence stiffness. This is due to

the shear lag, which influences the axial distribution in the columns where the stresses in the inner columns lag behind due to the bending of the spandrel beams.

In designing this type of building, an engineer should limit the level of shear lag and aim for structural behaviour similar to that of cantilevers.

Braced tubes

Unlike framed tubes, a braced tube consists of widely spaced columns that are stiffened with the use of diagonal bracing. However, braced tubes are only used for structures with less than 60 storeys, because when height increases more sway will be observed, similar to MRF structures. The John Hancock Center in Chicago (as shown in Figure 3.6) is one famous example.

Tube-in-tube

This structure system is a combination of an outer framed tube with an inner core (can be a concrete core wall or steel braced tube) and is called a tube-in-tube structure. The inner braced core and the outer tube are connected through the floor diaphragm. They work together in resisting lateral loads.

The external tube holds much greater structural significance compared to the internal core due to the structural depth it achieves (Jayachandran, 2003). Figure 3.7 shows a usual tube-in-tube structure model, with inner core and outer framed tube modelled.

Fig. 3.6   John Hancock Center in Chicago.
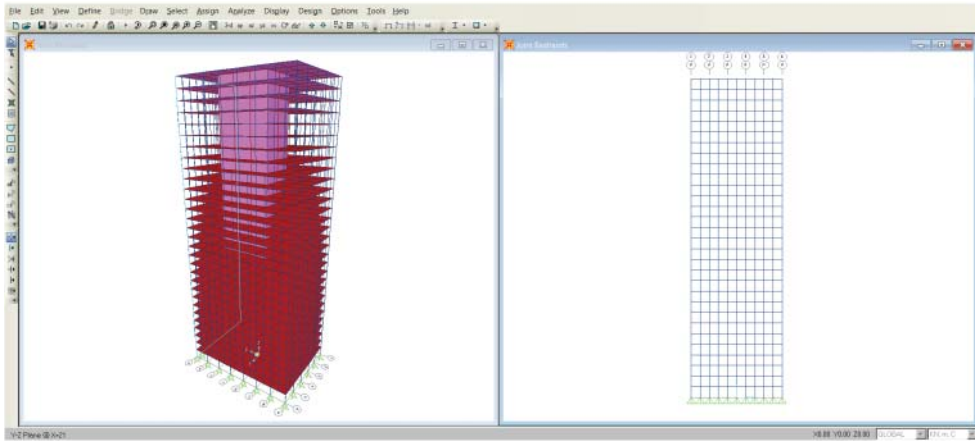
Fig. 3.7   Tube-in-tube structure model in SAP2000.
SAP2000 screenshot reprinted with permission of CSI.

Bundled tube

The bundled tube system is a further development of tube system, rather than using only one tube, several tubes are tied together to resist the lateral forces. One famous example is the Willis Towers (as shown in Figure 3.8). It consists of a set of singular tubes that are joined together forming a multi-cell tubed structure with individual

Fig. 3.8   The Willis Towers.

towers are connected by belt trusses. It is especially suitable in very tall structures. As the bundled tube structure has the ability to reduce shear lag, it hence provides a relatively lighter structure.

Figure 3.9 shows a detailed plan layout of the tower at each typical level. It can be seen that it consists of nine bundled tubes at the base, each approximately 23 m × 23 m. As the structure increases in height at several locations, two or more tubes are dismissed, with two tubes continuing to the top of the building from the 90th storey.

Modelling example of the Willis Towers using SAP2000

Here, we are going to demonstrate detailed steps of how to model the bundled tubes structure Willis towers in SAP2000.

Step 1: Set up a 3D model in AutoCAD

A 3D model including the frame element, such as beam, column and belt truss, is created in AutoCAD (as shown in Figure 3.10). The slab is not created in AutoCAD, as it is easier to create it later in SAP2000. In the experience of the author, one should divide the structure members into certain groups based on either different types of structural members or structural members in different storey, and save them into different layers. When importing to SAP2000, import them sequentially. This avoids chaos in the SAP2000 model and is demonstrated in Figure 3.11.

Step 2: Import the 3D model from AutoCAD into SAP2000

Figure 3.12 shows the 3D model after importing into SAP 2000. After importing the structural frame sequentially from different layers of AutoCAD, it is also convenient to group the members from each layer into the same group. This will facilitate the further modification of the member in the future. This is because, for complicated buildings such as the Willis Towers that consist of large numbers of structural members, it is hard to locate them efficiently without grouping them.

In the experience of the author, defining an efficient grid system is another effective way in modelling. This is because for a tall building an effective grid system will enable you to locate the structural member easily. As shown in Figure 3.13, a correct grid definition will enable you to access each floor plan or elevation easily, so you can define the slab at a certain level or modify a bracing in a certain elevation quickly in SAP2000.

Step 3: Analysis and result interpretation

After setting up the model, the load cases and load combinations will be defined, and the model will run under different load cases to get the results of displacement and internal force.

3.3.6 Diagrid structures and modelling example of the Gherkin

The diagrid system is derived from the conventional bracing system that consists of multiple diagonal grids positioned on the exterior of the building. Such systems exhibit greater structural effectiveness in comparison with the traditional diagonal bracing located within the cavity of central cores or in the facade. One famous project
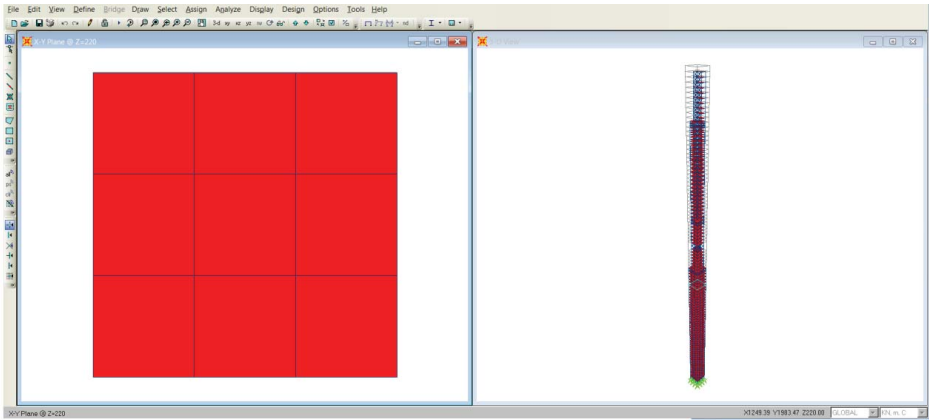
(a) Plan layout of Level 1–50
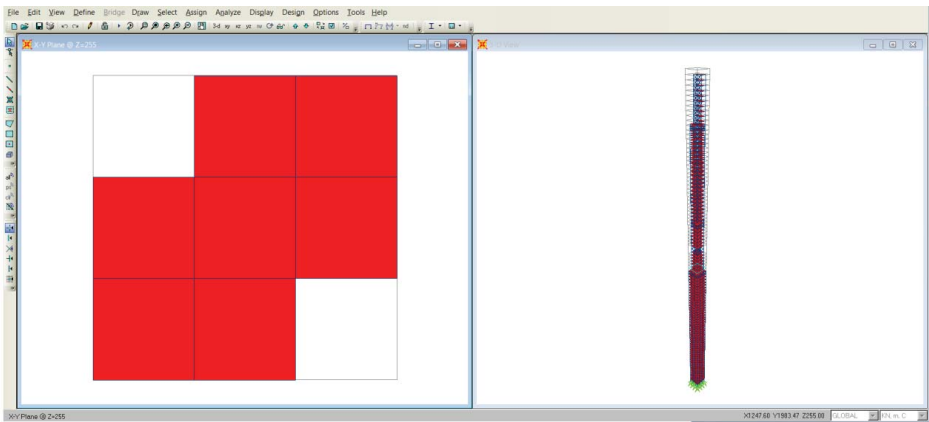
(b) Plan layout of Level 50–66
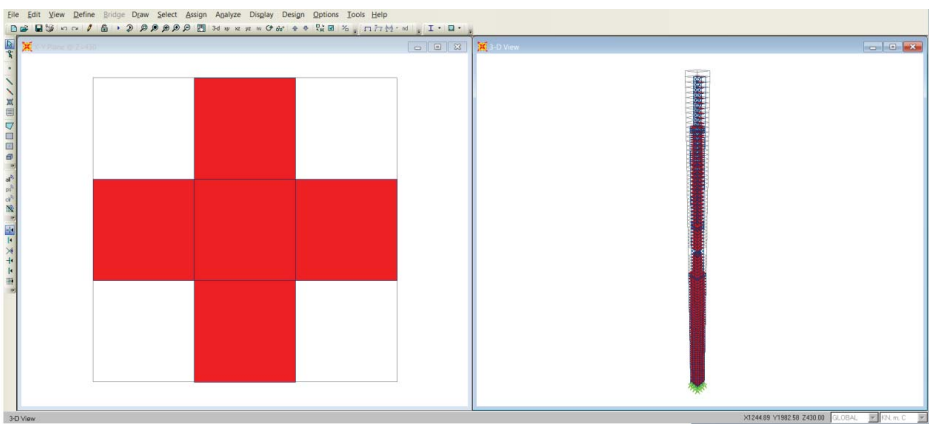
(c) Plan layout of Level 66–90

**Fig. 3.9   (a–d) Bundled tube system within the Willis Towers (SAP2000 model).
SAP2000 screenshot reprinted with permission of CSI.**

(d) Plan layout of Level 90–110

Fig. 3.9 (*continued*)
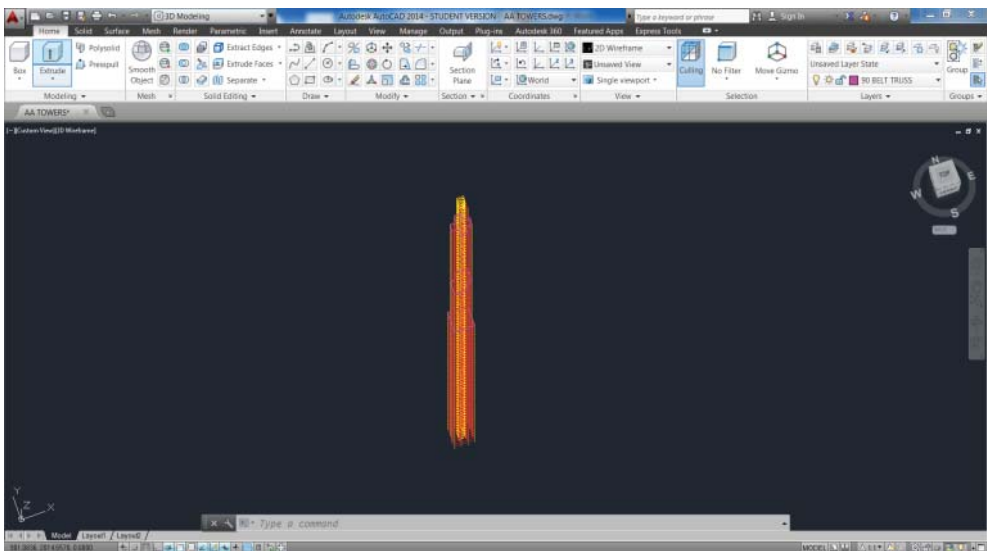
Fig. 3.10 **3D AutoCAD model of Willis Tower.**
AutoCAD.

example is the Gherkin in London. In this section, the modelling example of the Gherkin using SAP2000 will be demonstrated.

The triangular geometry of diagrid structures can effectively prevent structural failure as a result of lateral loads. Compared to conventional external bracing, it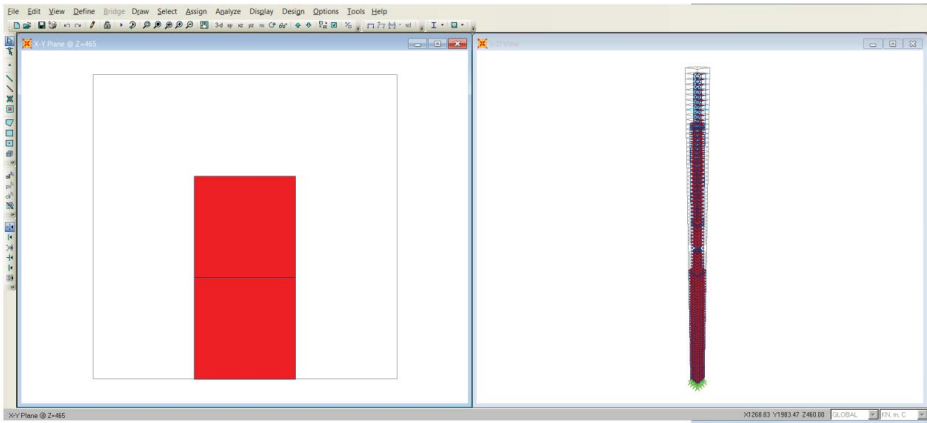 considerably minimises numbers of columns, especially less requirement for corner columns. Diagrid structures can also resist shear distortion through the axial resistance of diagonal members. The combination of an interior core and the external diagonal bracing provides extra stiffness, therefore enabling greater heights.

30 St Mary Axe, known as the Gherkin (as shown in Figure 3.14), is located in the heart of central London. The height of the building is 180 metres. The architect is
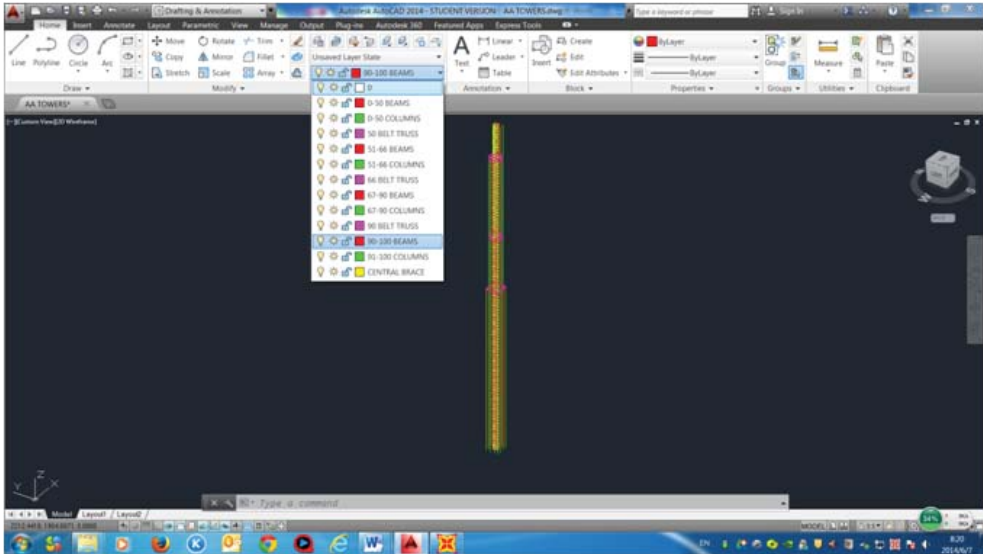
**Fig. 3.11   Arrange the structural elements in different layers in AutoCAD.
AutoCAD.**

**Fig. 3.12   3D SAP2000 model of the Willis Tower.
SAP2000 screenshot reprinted with permission of CSI.**

Fig. 3.13   Grid system in SAP2000.
SAP2000 screenshot reprinted with permission of CSI.
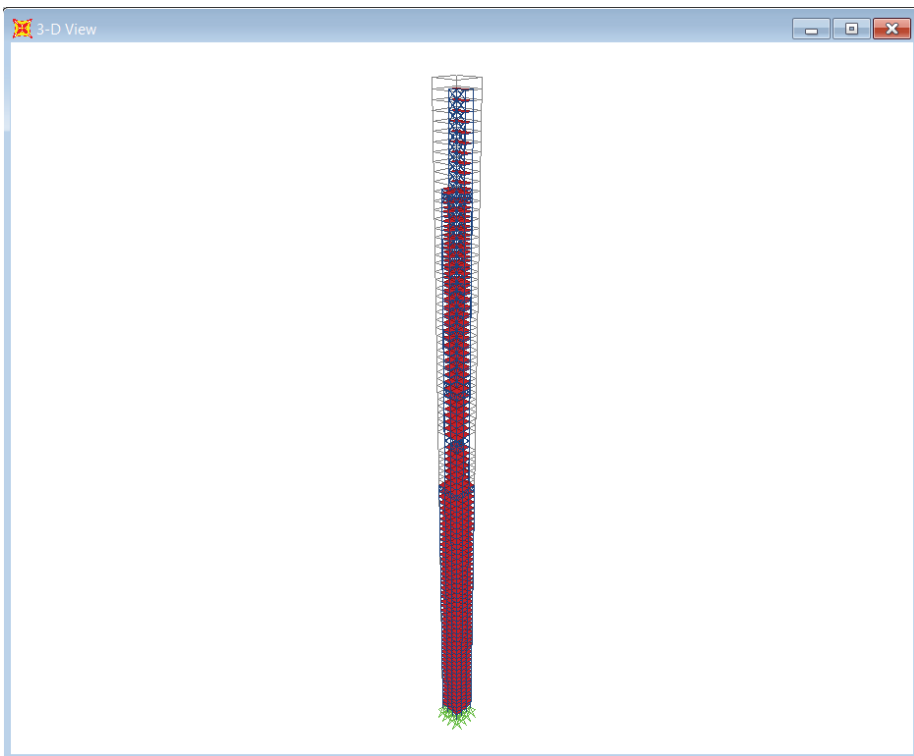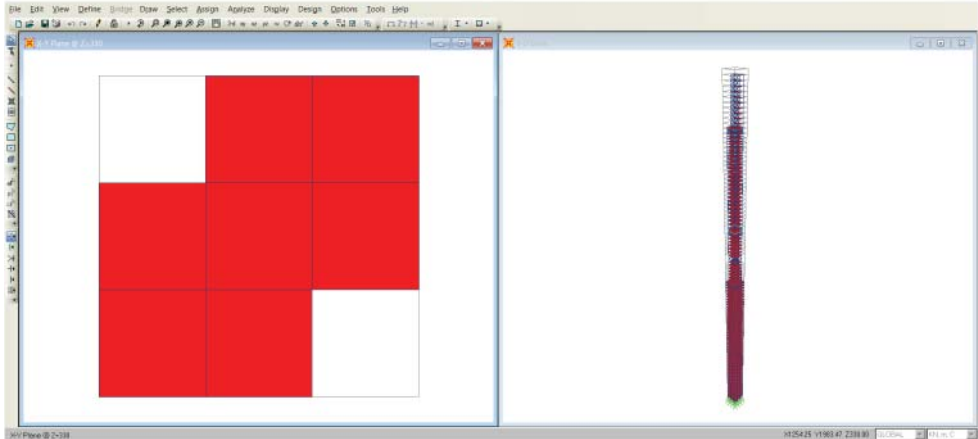

Fig. 3.14   30 St Mary Axe, London.
Photo from Structurae: International Database for Structural Engineering, courtesy of
Wilhelm Ernst & Sohn Verlag.

Foster & Partners. The construction of the Gherkin was started by Skanska in 2001 and was completed by 2003. 30 St Mary Axe was the first sustainable skyscraper constructed in London and was nominated for several building awards.

Due to its complexity, it is quite hard to set up the model directly from SAP2000. Therefore, the model is first set up in AutoCAD and then converted into SAP2000. In this section, how to set up the model will be introduced.

The model is set up based on some of the drawings of elevation and general arrangement of the structure provided by Foster + Partners, as shown in Figure 3.15. Using basic mathematics and the method of scaling, the diameter, height and the vertical as well as the horizontal cross-sections of the building were estimated. As the model is mainly for demonstration of the modelling techniques, it is not 100% identical to the original design. However, the main structural systems are similar, so it is sufficient to represent the original structure.

(a) Elevation
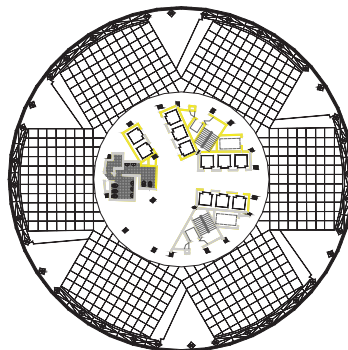
(b) Plan view

Fig. 3.15   Architectural drawings provided by Foster + Partners. (a) Image courtesy of BPR_Foster + Partners. (b) Image courtesy of Foster + Partners.

As the structural system of this building is very complicated, the model is divided into three main parts, which are: the slab system, the core system and the diagrid system. They are created in AutoCAD individually, with the same origin and unit, then converted into SAP2000 sequentially, as their origin and units are the same, so after the three-step conversion they will connected to each other in SAP2000. A 3D grid system is also defined along the height of the building in SAP2000. Therefore, the engineer can access the plan view of each floor to make further modifications, such as for the beam and slabs.

The basement of 30 St Mary Axe is not taken into consideration, thus the model starts from the ground floor up to the 40th floor.

Step 1: Set up the slab geometry

The layout of each floor was first created in AutoCAD. Figure 3.16 depicts the plan layout of the 1st floor. Figure 3.17 shows the 3D view of the slab edges. Remember,
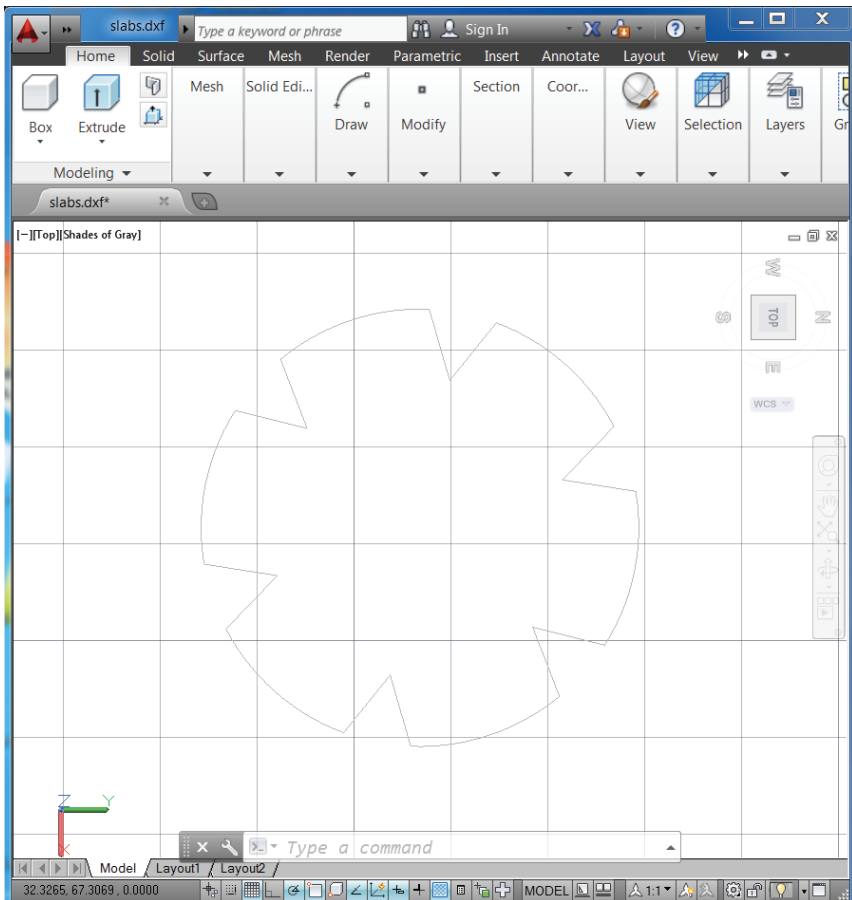

Fig. 3.16   Plan layout of the slab in AutoCAD.
AutoCAD.
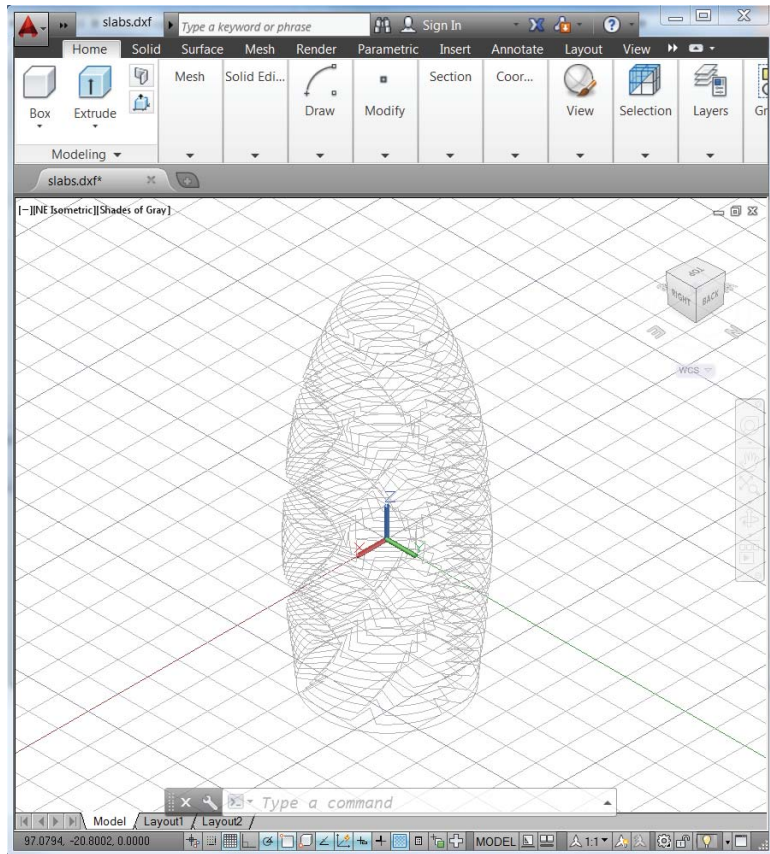
Fig. 3.17   3D model of slab edges in AutoCAD.
AutoCAD.

only the edges of the slabs are created in AutoCAD, the slabs are not physically created; as mentioned earlier, it is more efficient to create the slabs in SAP2000. Finally, the slabs edges were imported from AutoCAD into SAP2000 as shown in Figure 3.18. In AutoCAD, one should make sure that the lines are polylines. The curves can be approximated using the polyline segments.

Step 2: Set up the steel core

From further investigation of the photos on the website, it can be seen that the building has two cores. The outer core of the structure has a cylindrical shape while the inner core has a trapezoidal shape. Both cores use double diagonal bracing to increase the stiffness of the structure, resulting in better resistance to lateral loading.

The inner and outer cores of the structure were created first in AutoCAD as shown in Figure 3.19 and 3.20, based on the pictures. The outer core has two double diagonal bracings in each floor forming a spiral shape, while the inner core has one double diagonal bracing for each floor forming a spiral shape as well.

Fig. 3.18   Slab edge imported into SAP2000.
AutoCAD.

Fig. 3.19   Inner core created in AutoCAD (without the beams showing).
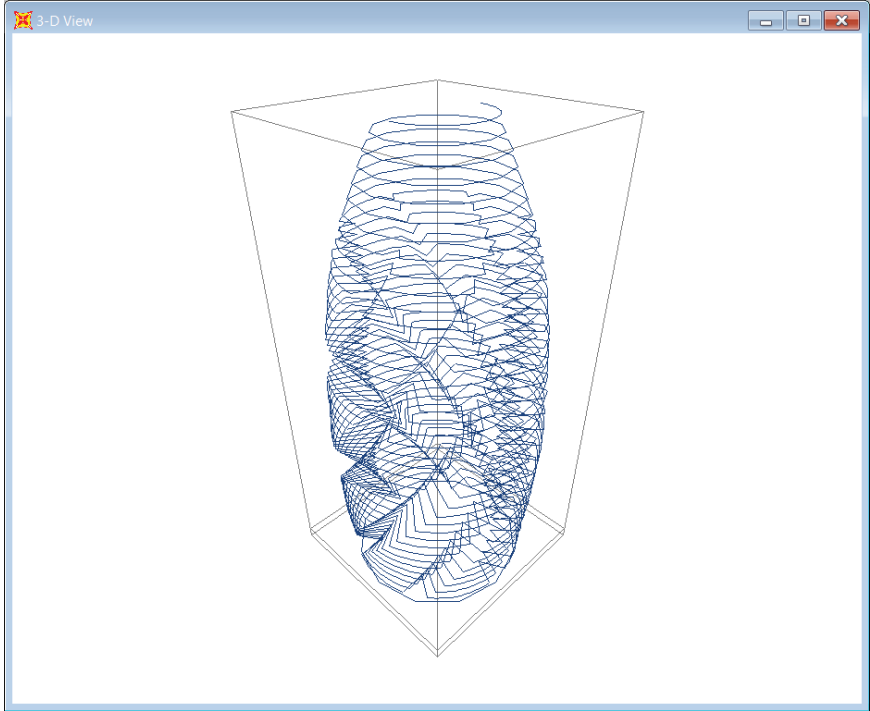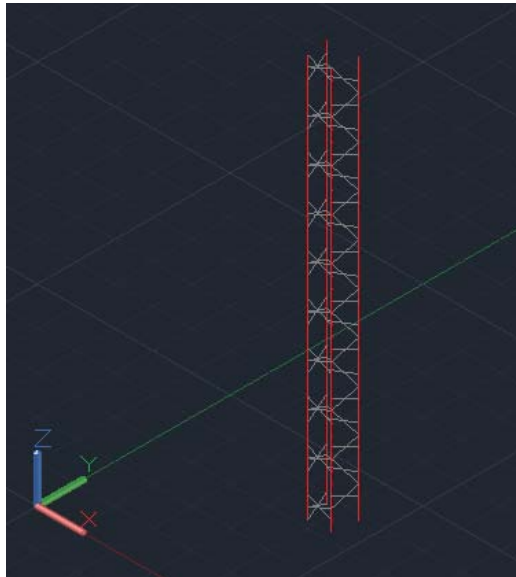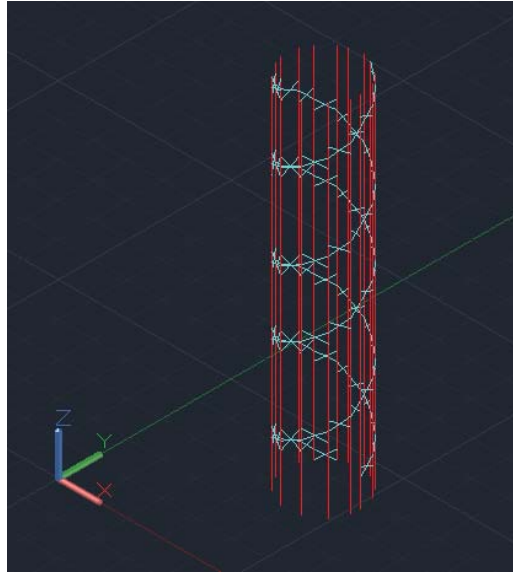**AutoCAD.**

**Fig. 3.20   Outer core created in AutoCAD (without the beams showing).
AutoCAD.**

Step 3: Setting up the diagrid system

The diagrid system consists of tubular sections forming a rhomboidal shape and then divided into sub-triangles with I-beam sections. The tubular and I-section struts were covered with plates to form a triangular shape for aesthetic reasons and better values of fireproofing resistance.

The diagrid system is also created in AutoCAD as shown in Figure 3.21: here the structural members are represented by polylines that are divided into certain groups and denoted using different colours and saved into different layers of AutoCAD. In the author's experience, using different colors means one can easily identify the different groups of lines and modify them in AutoCAD if necessary. As these lines are saved in different layers, it is therefore easy to switch off and on the corresponding layers to allow easy access to them.

The model is then converted into SAP2000 as shown in Figure 3.22.

Step 4: Rationalisation of the model in SAP2000

After being imported into SAP2000, a further rationalisation of the model needs to be conducted as some structural members may not be connected in the AutoCAD drawings, thus manual adjustment is also carried out to make sure the model is well connected.

In certain situations, for the connectivity of joints after importing into SAP2000, all joints can be merged using *Edit – Edit Points – Merge Joints* with an accuracy of 0.01 m tolerance or any other suitable tolerance depending on the dimensions of the model. One of the disadvantages of merging the nodes is that it may make the structure distorted. Therefore one should be careful when using this option.
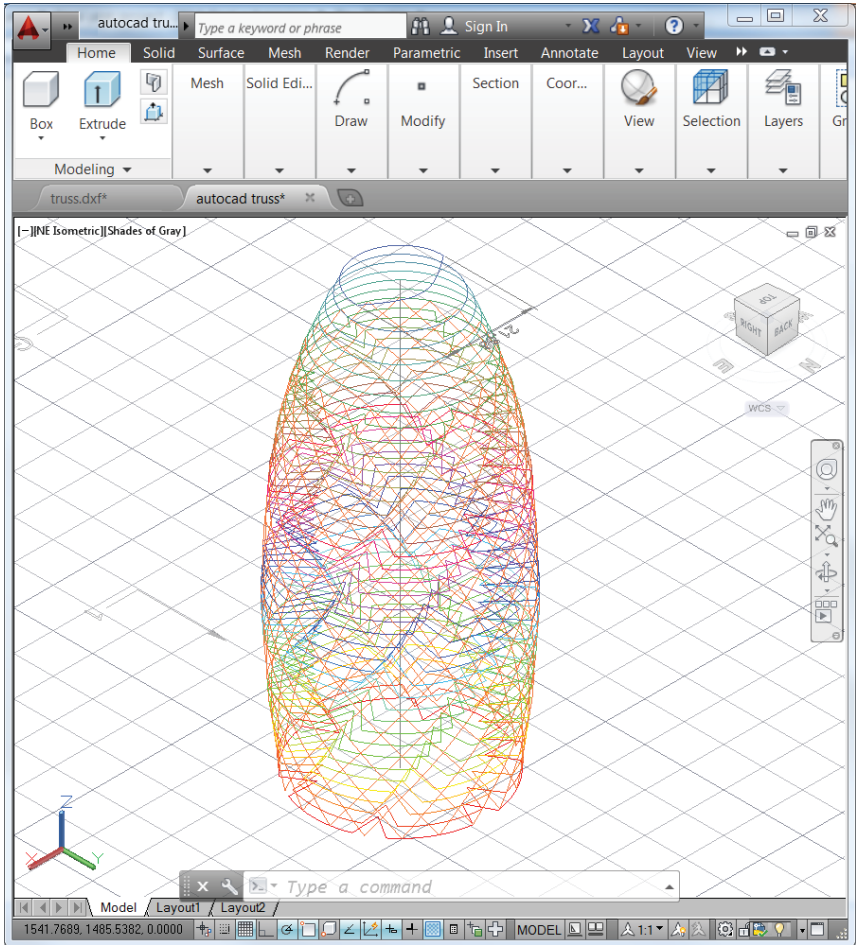
Fig. 3.21   Diagrid system created in AutoCAD.
AutoCAD.

Step 5: Define the load

The load case and load combination will be defined in this step, and the model will run under the different load cases to get results of displacement and internal force. It worth noting that, as will be introduced in another modelling case of Buji Kalifa for wind load, the diaphragm of each floor needs to be defined and assigned a wind load pattern in SAP2000. Here, as the shape of the building is quite special, the wind load was worked out manually according to Eurocode (2004) and then applied as point loads on each floor (as shown in Figure 3.23, it can also be seen in Figure 3.22). The dead and live load are also worked out according to Eurocode (2005).

Step 6: Analysis

After the model has been set up, analysis will be performed and results such as the lateral displacement and stress can be extracted from the model, which will be introduced in Chapter 4.

Fig. 3.22   3D model in SAP2000 after 3-step importation.
SAP2000 screenshot reprinted with permission of CSI.

3.3.7   Super frame (mega frame) structures and modelling example

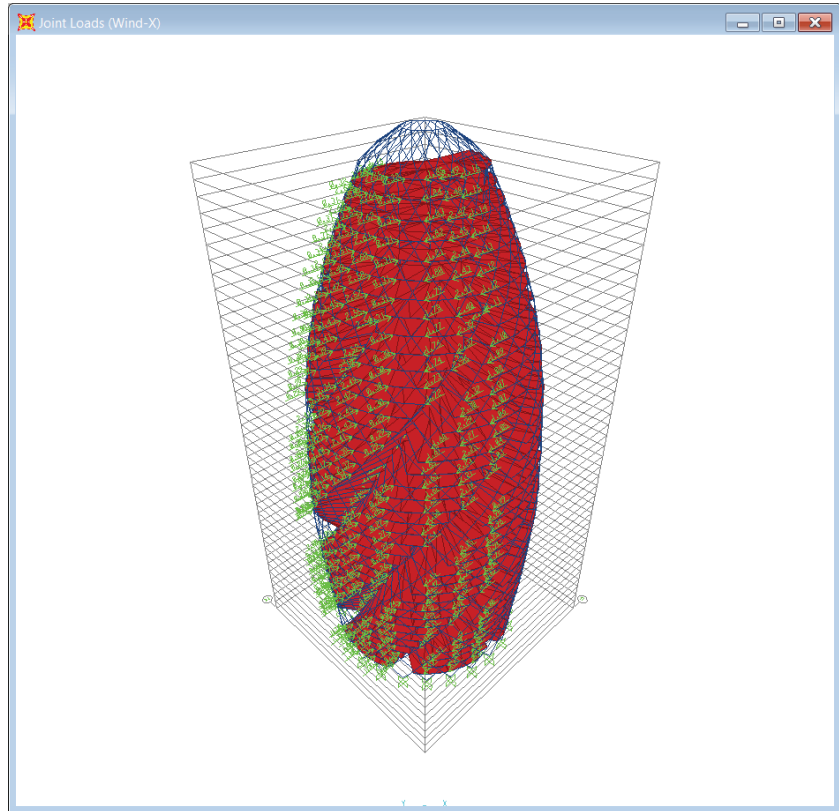Another important lateral stability system to be introduced here is called the super frame (or mega frame) structure. It consists of mega columns and mega girders which are the main stability system of the building. They are connected together at approximately 10 to 25 storeys, which work together as the primary lateral resisting structural systems. The secondary or sub-structures are connected to the super-structures. One famous example is the HSBC headquarters in Hong Kong, as shown in Figure 3.24.

Figure 3.25 and 3.26 show the modelling example of a mega frame structure using SAP2000, which illustrates the major structural frame of this type of building.

3.4   Modelling example of the Burj Khalifa

The Burj Khalifa (shown in Figure 3.27) is currently the tallest building in the world with a height of 828 m. It consists of approximately 160 floors. As shown in Figure 3.28, which is the general arrangement of a typical floor of the building, the Burj Khalifa is strengthened at its three wings by high performance concrete walls, a structural system that is an evolution of the simple shear wall core referred to as a 'buttressed' core.
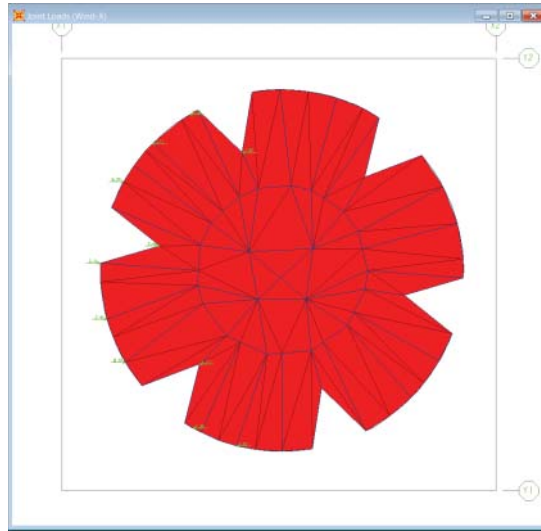
(a)
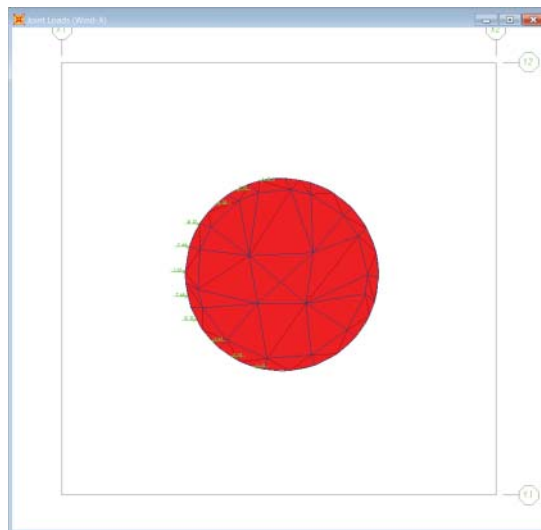
(b)

Fig. 3.23 **(a and b) Wind load applied on different floors.**
SAP2000 screenshot reprinted with permission of CSI.

The three wings are strengthened through a central core of hexagonal geometry, providing torsional resistance. The walls that spread across each wing from the central core and increase in dimension at the end of each corridor act in a similar manner to a beam, which enables the structure to combat the effects of wind loads. The system is highly effective for achieving an extreme height.

Fig. 3.24   HSBC headquarters in Hong Kong.
Photo from Structurae: International Database for Structural Engineering, courtesy of Wilhelm Ernst & Sohn Verlag.

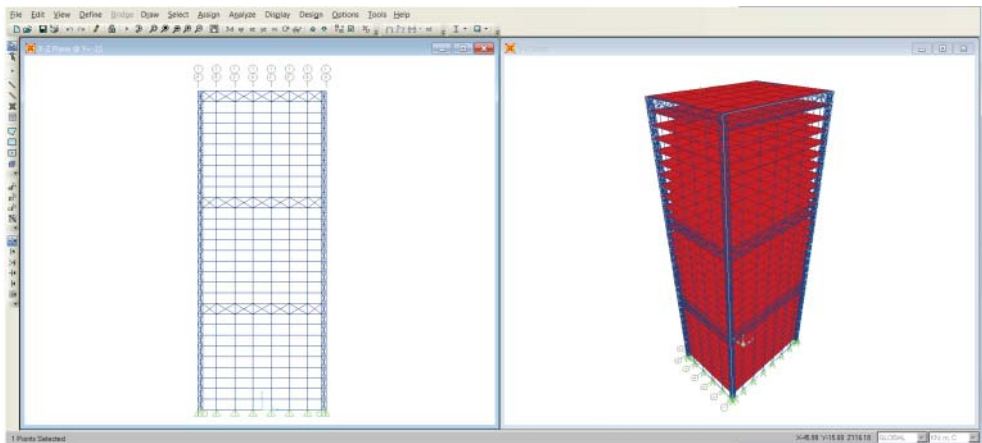**Fig. 3.25   Super frame (mega frame) structures (elevation and 3D view).**
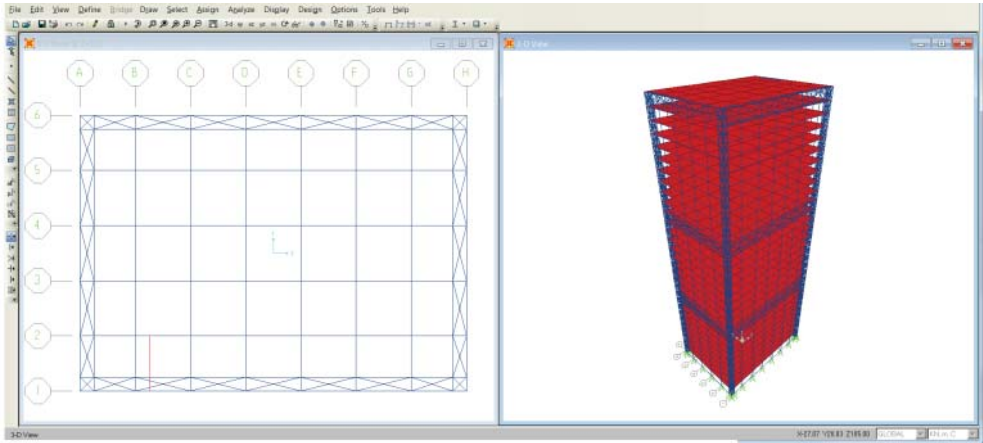**SAP2000 screenshot reprinted with permission of CSI.**

Fig. 3.26   Super frame (mega frame) structures (plan and 3D view).
SAP2000 screenshot reprinted with permission of CSI.

Fig. 3.27   The Burj Khalifa.
Photo from Structurae: International Database for Structural Engineering, courtesy of
Wilhelm Ernst & Sohn Verlag.

R/C Hammer head Wall
[1300 mm]

R/C Nose columns
[1500 mm]

R/C Corridor shear wall
[650 mm]

Edge of R/C
flat plate

R/C Perimeter column
[3500¡ 600 mm]

R/C Link
beamt

R/C Hexagonal core wall
[600 mm]
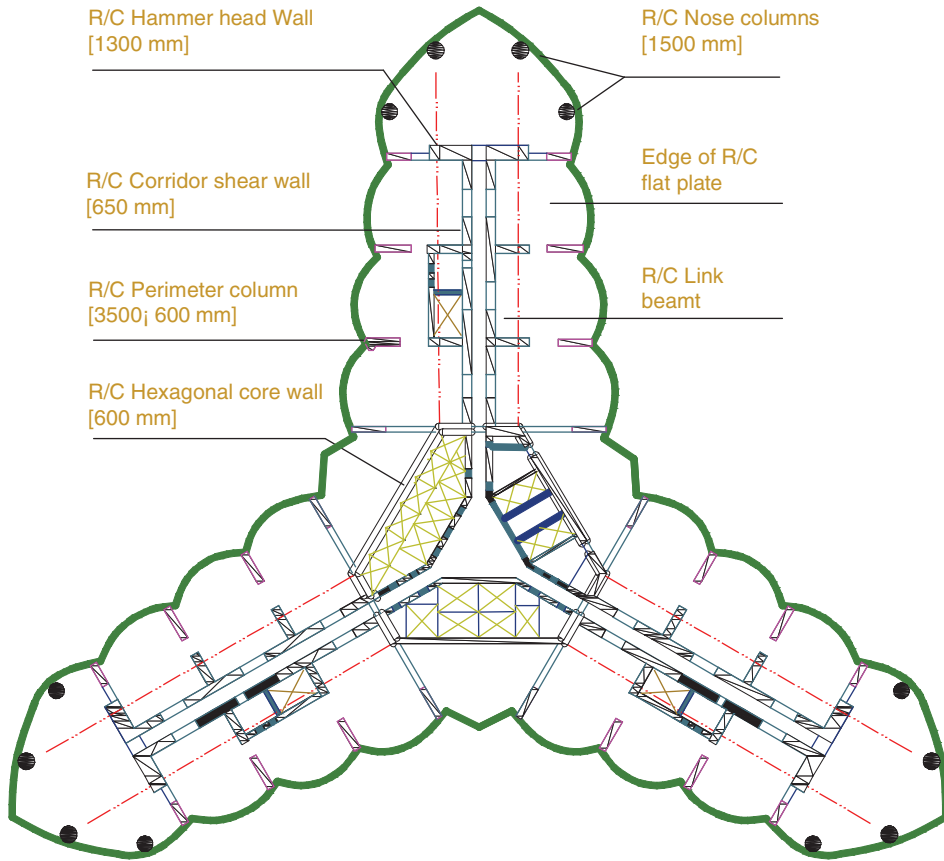
Fig. 3.28   Structural systems of the Burj Khalifa Abdelrazaq, 2011.

The Burj Khalifa consists of only a few columns, positioned at its boundary. The building's columns are interconnected using outriggers, which maximises the structure's stiffness (Abdelrazaq, 2011).

The above structural system provides an effective system to resist both vertical and horizontal loading for such a tall structures.

3.4.1   Model set up

In this section, the Burj Khalifa is modelled using the SAP2000 program; the modelling technique will be introduced in detail. The model is built based on the plan and elevation drawings and photos on the web; as the purpose of this book is to demonstrate the means of modelling, 100% accuracy in representing the original project cannot be guaranteed.

The Burj Khalifa has a complex 'Y' shaped structure, as shown in Figure 3.28, which is based on the hymenocallis flower. To model such a complicated structure on SAP2000 would be extremely difficult, therefore the model was created in AutoCAD. The 3D model from AutoCAD was imported into SAP2000 for analysis.

Step 1: 3D model set up using AutoCAD

Similar to other modelling cases, the wireframe of the structure was created in Auto-CAD as shown in Figure 3.29. Figure 3.30 demonstrates the procedure for setting up the model from the bottom floor to the roof level truss. Remember, here, only the layout of each floor, column and beam and roof level truss is modelled, the shell elements such as the slab and shear walls will be defined in SAP2000. However, the control points, such as the location of the shear wall and openings, are also constructed in this model. Similar to other models demonstrated in this chapter, make sure different structural element are put into different layers.

Step 2: Modelling using SAP2000

After the 3D model has been set up in AutoCAD, the geometry from each layer will be imported into SAP2000 sequentially, and the model built in SAP2000 as shown in Figure 3.31 and 3.32.

Step 3: Loading

The loading, such as dead, live and wind, must then be defined. Here, we will introduce the method for defining the wind load. Wind loads on the structure were selected from the Eurocode 1 Part 1–4 (2005) code of practice, which is available from SAP2000. In reality, a specialist in wind design was employed and several wind tunnel tests were carried out, and this covered all the variables affecting the building. As this book concentrates on introducing the means to model a complex structure, some reasonable simplification has been made.

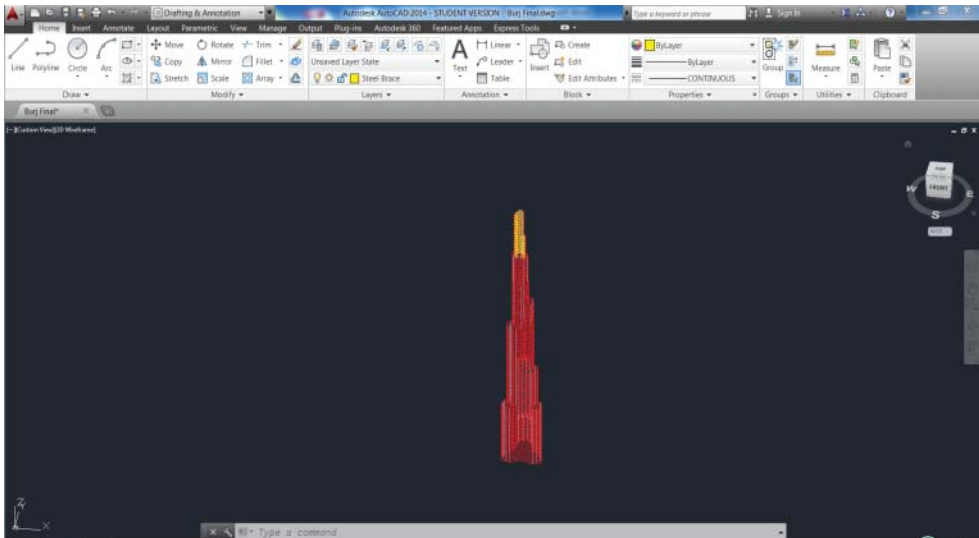**Fig. 3.29   3D model in AutoCAD.**
AutoCAD.
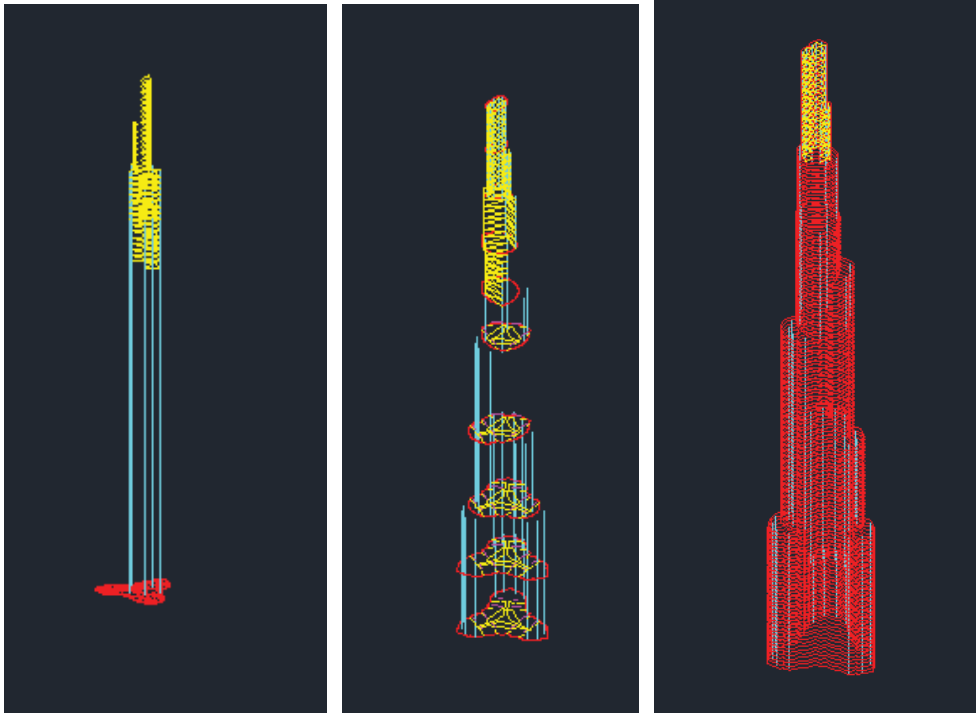
Fig. 3.30   Procedure of the model set up in AutoCAD.
AutoCAD.
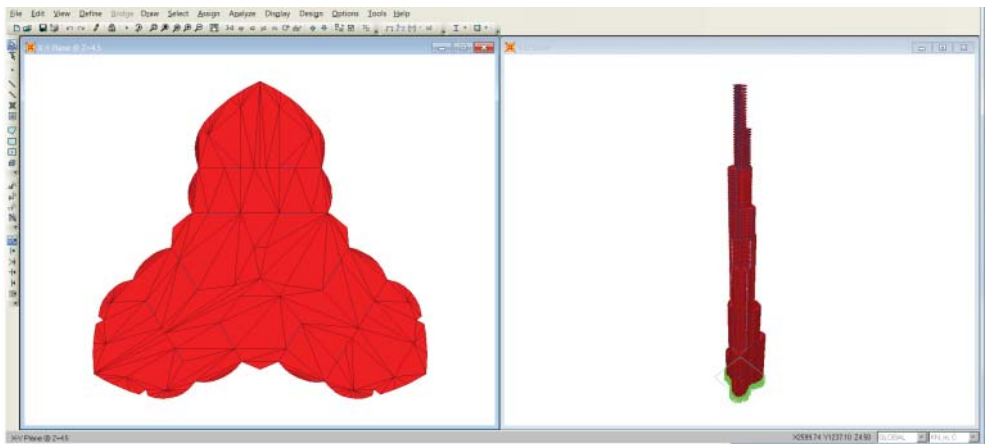
Fig. 3.31   3D model in SAP2000 (floor plates shown in the left window).
SAP2000 screenshot reprinted with permission of CSI
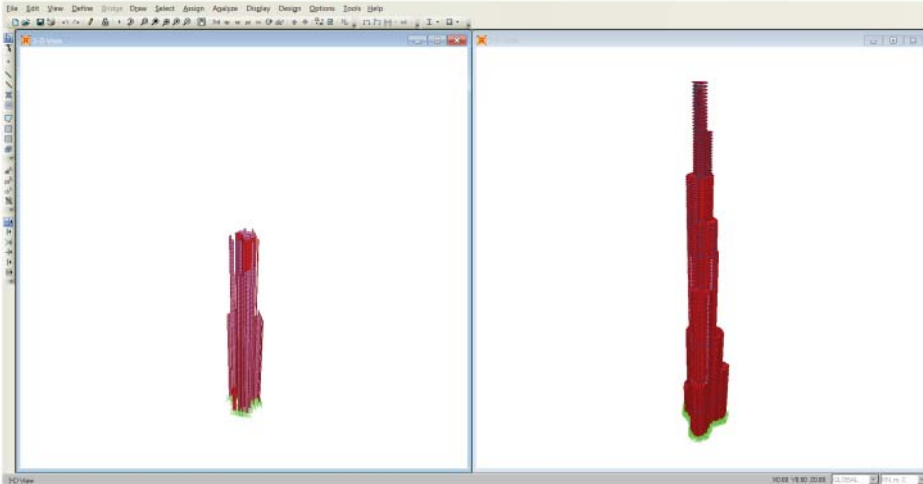
Fig. 3.32   Centre core wall and shear walls.
SAP2000 screenshot reprinted with permission of CSI.

Define diaphragm

- From the ribbon, click *Select – All*, the whole structure including all the nodes are selected.
- Then select *Assign – Joint – Constraint*.
- A new window will pop up, as shown in Figure 3.33, select *Diaphragm* from *Constraint Type*.
- Then click *Add New Constraint*. A new window will pop up, make the selection as shown in Figure 3.34 and click *OK*. The diaphragm has been assigned to the nodes on each floor.
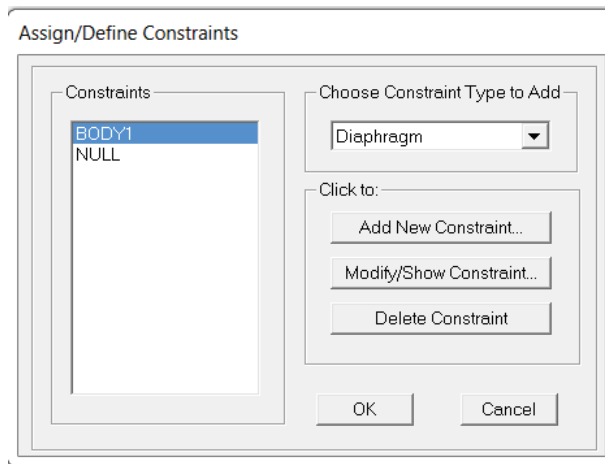
Fig. 3.33   Defining the diaphragm.
SAP2000 screenshot reprinted with permission of CSI.
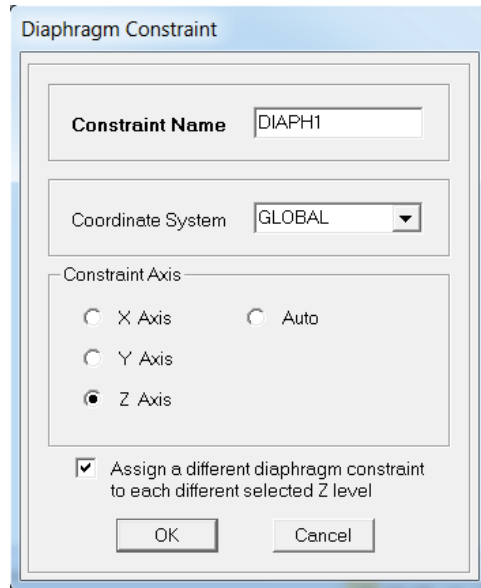
Fig. 3.34   Defining the diaphragm parameters.
SAP2000 screenshot reprinted with permission of CSI.

Define wind load pattern

- In the ribbon, click *Define – Load Patterns*, a new window will pop up.
- Define *Dead* and *Live* load case, then, define *Wind* load case – in *Auto Load Pattern* choose Eurocode1 2005, see Figure 3.35.
- Click *Modify Lateral Load Pattern*. A new window will pop up, as shown in Figure 3.36.

The figure above shows the wind pattern applied on the structure. The wind velocity applied is 70 m/s; this is selected based on the general wind speed around that region.

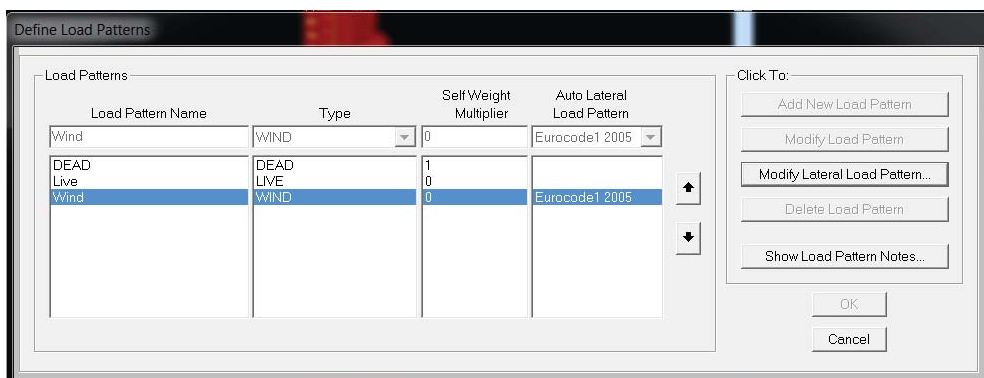**Fig. 3.35   Defining the load pattern.**
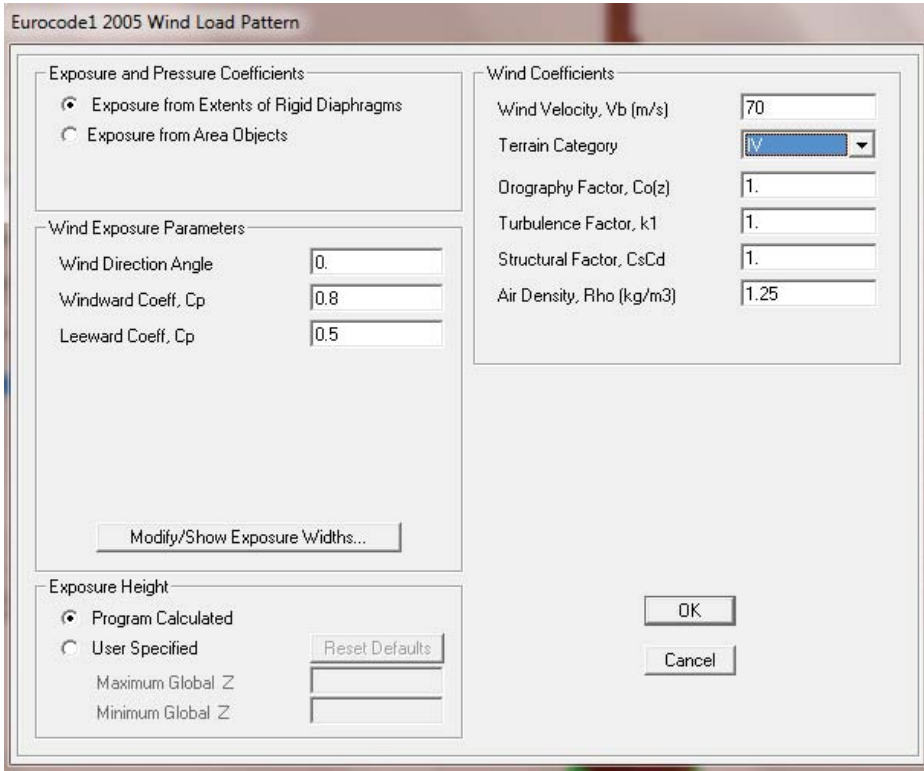**SAP2000 screenshot reprinted with permission of CSI.**

Fig. 3.36   Defining the load pattern parameters.
SAP2000 screenshot reprinted with permission of CSI.

It can be more severe but this is an average wind load that is applied. The Burj Khalifa is built in the city, so therefore category IV is selected. The terrain around the building is covered with buildings and the buildings' average height exceeds 15 m. The other parameters are selected after researching the wind loads on buildings.

- Click on *Modify* and *Show Exposure Widths*, the diaphragm defined on each floor is displayed as shown in Figure 3.37.
- SAP2000 will work out the resultant force of the wind on each floor base on the parameters input to the model and apply them at the shear centre of the diaphragm.
- To get a more accurate value, the reader can also choose the *User Defined Option*, and work out the diaphragm in a spreadsheet and import it into SAP2000.

3.4.2   Analysis and result

After the model is set up, a detailed analysis will be performed. The results, such as the lateral deflection and stress, will be extracted from SAP2000 through *Display – Show Table – Joint Output-Displacement*. The inter-storey drift can be worked out through exporting the result table into Excel and carrying out further processing. The detailed methods for extracting the results will be demonstrated in the next chapter.

Fig. 3.37 **The diaphragm displayed in SAP2000.**
SAP2000 screenshot reprinted with permission of CSI.

3.5 Modelling example of Taipei 101 with tuned mass damper (TMD)

Taipei 101 (as shown in Figure 3.38) held a record height of 508 m. It is a building that has a number of various lateral stability systems, such as outriggers, moment resisting frames and super columns made from high strength concrete and steel. In order to provide the maximum comfort to occupants, Taipei 101 comprises the largest tuned mass damper (TMD) in the world. With its phenomenal height, it successfully stands up to the two most destructive adversaries of tall buildings: earthquakes and typhoons.

Similar to that introduced in the previous sections, the model is set up in SAP2000 as shown in Figure 3.39. The feature to be introduced here is how to simulate the TMD in a tall building.

3.5.1 TMD modelling

In order to model the damping of the TMD, a nonlinear link was used in SAP2000 to simulate the TMD, which had the relevant stiffness and mass assigned to it and was connected to a rigid joint as shown in Figure 3.40.

The 730 000 kg mass suspended from the 92nd to the 88th floor in Taipei 101 is modelled using a mass that is attached to a fixed joint and then connected to the 92nd floor of the 3D model by a spring of equivalent stiffness and a dash point connected in parallel.

Fig. 3.38   Taipei 101.
Photo from Structurae: International Database for Structural Engineering, courtesy of Wilhelm Ernst & Sohn Verlag.

The natural frequency and the equivalent stiffness are worked out as follows:

The 1st mode or the natural period of the pendulum was calculated using the simple formula below:

$$T_\mathrm{d} = 2\pi \sqrt{\frac{l}{g}} = 2\pi \sqrt{\frac{11.5}{9.81}} = 6.802 \text{ sec onds} \tag{3.2}$$

Where:

T_d = natural period of pendulum in seconds
l = length of suspended pendulum in metres
g = acceleration due to gravity in m/s^2
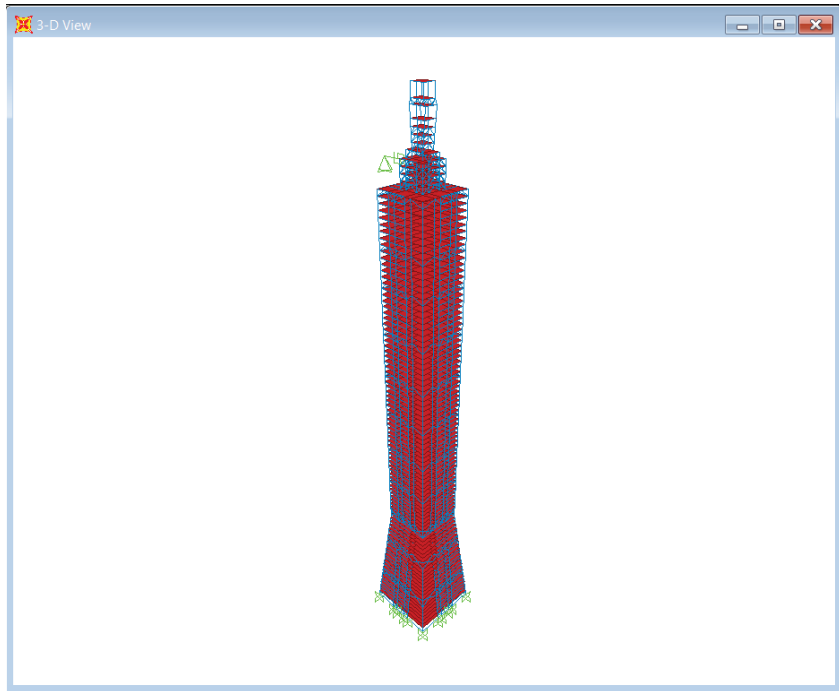
Fig. 3.39   Model of Taipei 101 in SAP2000.
SAP2000 screenshot reprinted with permission of CSI.
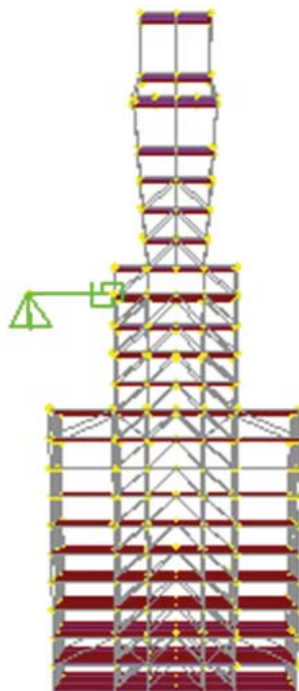
Fig. 3.40   Nonlinear link to model the TMD in the Taipei 101 model.
SAP2000 screenshot reprinted with permission of CSI.

The natural period of the TMD allowed the model to be tuned to the relevant frequency.

The equivalent stiffness of damper in the SAP2000 model was also calculated using the simple formula below:

$$k_{eq} = \frac{w_d}{l} = \frac{(730\,000 \times 9.81)}{11.5} = 622\,721.7391\,\text{N/m} \tag{3.3}$$

Where:

K_{eq} = Equivalent stiffness of damper in N/m
w_d = Weight of damper in Newtons
l = length of suspended pendulum in metres

The method to define the link is briefly shown here, for a detailed example please refer to Chapter 10 and the modelling example of vibration analysis of the Millennium Bridge.

- Click on *Define* and *Section Properties*, then *Link/Support Properties*; a new window will pop up, as shown in Figure 3.41.
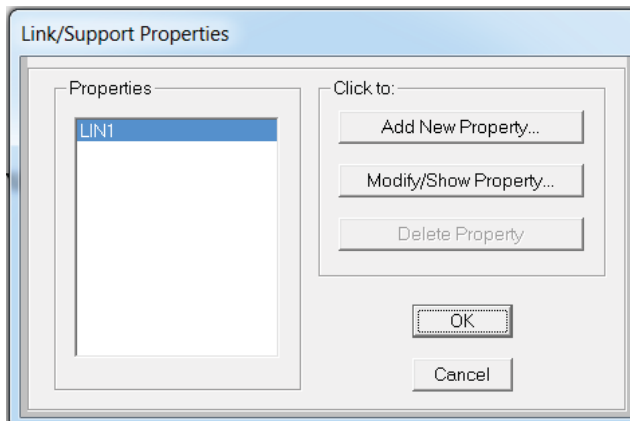
Fig. 3.41 Define link properties.
SAP2000 screenshot reprinted with permission of CSI.

Fig. 3.42 Selection of link element.
SAP2000 screenshot reprinted with permission of CSI.

- Click on *Add New Property* to define the property of the link.
- Click on *Draw*, then *Draw Two Joint Link*. A new window will pop up, as shown in Figure 3.42. Choose the link element we just defined.
- Draw the link between the fixed points and the centre of storey 92 in the model.
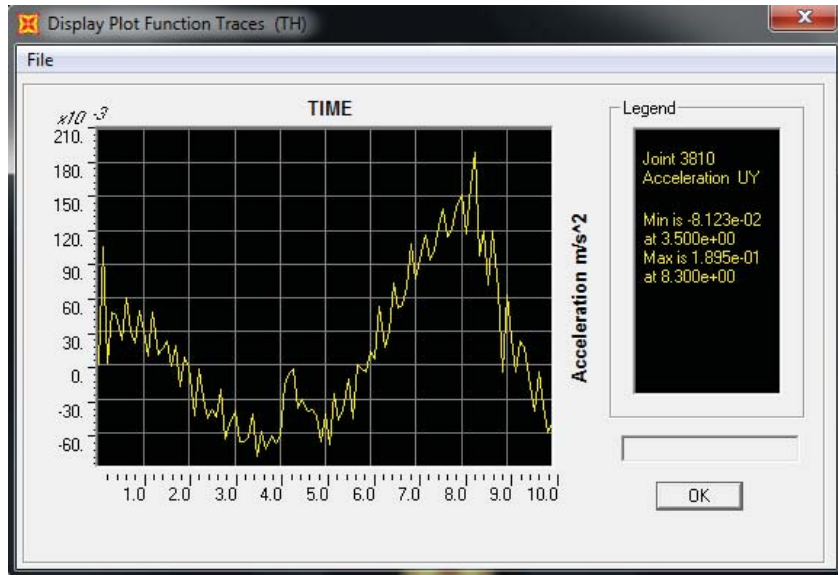
Fig. 3.43   Time history result without TMD.
SAP2000 screenshot reprinted with permission of CSI.
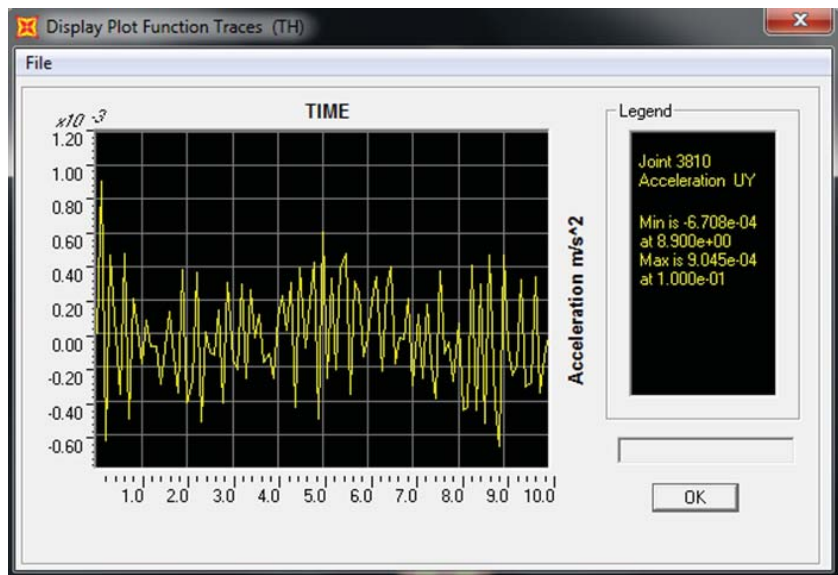
Fig. 3.44   Time history result with TMD.
SAP2000 screenshot reprinted with permission of CSI.

3.5.2   *TMD modelling result*

In order to assess the effectiveness of the TMD modelling, a time history analysis for both models consisting of the TMD and a model without a TMD was conducted. The results are shown in Figure 3.43 and 3.44. From the comparison, it can be seen that, for the model with TMD, acceleration reduced dramatically, which satisfies the relevant comfort criteria given by the codes of practice.

3.6   Conclusion

In this chapter, structural systems – especially the lateral resisting system – were introduced and demonstrated through the modelling of several existing tall building projects. Further introductions of tall building design can be found in Taranath (1998), Nair (1998) and Smith and Coull (1991).

References

Abdelrazaq, A. 2011. "Validating the Structural Behaviour and Response of Burj Khalifa" Council on Tall Buildings and Urban Habitat Journal Issue II http://technicalpapers.ctbuh.org.

Ali Mir M. and Moon K. S., 2007. "Structural developments in tall buildings: current trends and future prospect", *Architectural Science Review*, 50, 3: 205–223.

Jayachandran P., 2003. *Design of Tall Buildings: Preliminary Design and Optimisation*. Coimbatore, India: PSG College of Technology.

Kowalczyk R., Sinn R, Kilmister M. B. 1995. *Structural Systems for Tall Buildings*. New York: McGraw-Hill.

Nair R.S. 1998 "Belt Trusses and Basements as 'Virtual' Outriggers for Tall Buildings", *Engineering Journal /Fourth Quarter:* 140–146.

Narim, F. 1989. *Design for Drift and Lateral Stability, Seismic Design Handbook*, New York: Van Nostrand Reinhold.

Smith, B.S., Coull, A. 1991. *Tall Building Structures, Analysis and Design*, John Wiley & Sons, INC.

Taranath B.S. 1998. *Structural Analysis & Design of Tall Buildings*. New York: Mc Graw Hill.

Weingardt, R. 2005. *Engineering Legends*. ASCE Publications. pp. 76.

Codes and regulations

Eurocode 1 Part 1-1 2001: Actions on structures - General actions Densities, self-weight, imposed loads for buildings. Commission of the European Communities.

Eurocode 1 Part 1-4 2005: Actions on structures - General actions Wind actions. Commission of the European Communities.

4 Earthquake analysis of buildings

4.1 Introduction

In current global design practice, a large majority of projects are located in either high or mediate seismic activity zones such as Japan, California, Greece and Turkey; therefore, earthquake design is another important skill for structural engineers. In this chapter, the basic earthquake design principles will be introduced along with the different earthquake analysis methods such as push-over analysis, response spectrum analysis and time history analysis that are widely used in current design practice. The relevant earthquake design codes will also be introduced.

4.2 Basic earthquake knowledge

Earthquakes are caused by the rupturing of faults in the earth's crust, landslides and volcanic eruptions. Man-made activities such as a large explosion can also cause an earthquake. Normally, earthquakes are located close to the borders of tectonic plates. These plates are constantly in motion so stresses build up and release energy through the earth's crust, causing the shaking of the ground.

4.2.1 Categories of earthquake waves

The released energy during an earthquake is transferred in the form of waves. These waves can be divided in two basic categories: body waves and surface waves.

Body waves are transferred through the inner part of the earth which consist of primary waves (P) and secondary waves (S). Primary waves are also called compressional waves, and travel in longitudinal direction, push and pull the earth and are faster than S-waves. Secondary waves are also called shear waves and have a transverse direction, which shakes the earth vertically and horizontally. Their motion is critical and can cause damage to structures. Also their speed is slower than P-waves.

The other part of a seismic wave is the surface wave, which separates into Love and Rayleigh waves. They travel on the earth's surface; both have slower speed than body waves and they generally cause the worst damage in structures.

4.2.2 Measurement of earthquake

There are two ways that an earthquake can be measured: the Richter magnitude and the modified Mercalli intensity. In current design practice, the Richter scale is widely used to measure the magnitude of an earthquake. The Richter scale is a logarithmic expression of the magnitude of energy, which defines magnitude as the logarithm of the ratio of the amplitude of the seismic waves to arbitrary, minor amplitude.

The modified Mercalli intensity scale measures the potential damage of an earthquake. The range of this scale depends on observations of people's reactions during an earthquake, the damage to structures and on physical effects caused by an earthquake.

4.3 Basic dynamic knowledge

This section outlines basic dynamic analysis knowledge, such as the governing equations of motion. It starts with the response of a single-degree-of-freedom system (SDOF), and then a multi-degree-of-freedom (MDOF) system will be introduced. Finally, the response spectrum will also be introduced.

In general, there are four basic analytical models for dynamic problems, they are: SDOF, MDOF, the continuous system and generalized SDOF. However, in earthquake engineering, SDOF and MDOF are the most widely used analytical models. Therefore, a detailed introduction will be given in following sections.

4.3.1 SDOF

An analysis of the dynamic system can be simplified by assuming the mass of the structure to be concentrated at discrete points, which greatly simplifies the analytical problem because inertial forces develop only at these mass points. In this case, it is necessary to define the displacements and accelerations only at these discrete locations.

The simplest dynamic system is the one-degree-of-freedom system, where a concentrated mass is constrained to move in one direction only, and so can be defined in terms of a single coordinate (x).

An example of SDOF can be simplified to a single mass (M) attached to a spring (k) and a damper (c) that is subjected to an applied force $F(t)$. We call it Mass–Spring–Damper System as shown in Figure 4.1.

The governing equation of motion of the SDOF can be expressed as

$$m\ddot{x} + c\dot{x} + kx = F(t) \tag{4.1}$$

Where:
$m\ddot{x}$ is the inertial force
$c\dot{x}$ is the damping force, and c is the damping coefficient
kx is the elastic force
$F(t)$ is the external dynamic force

Further explanation of SDOF can be referenced in Clough and Penzien (1993).
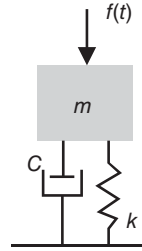
Fig. 4.1   SDOF Mass–Spring–Damper System.
AutoCAD.
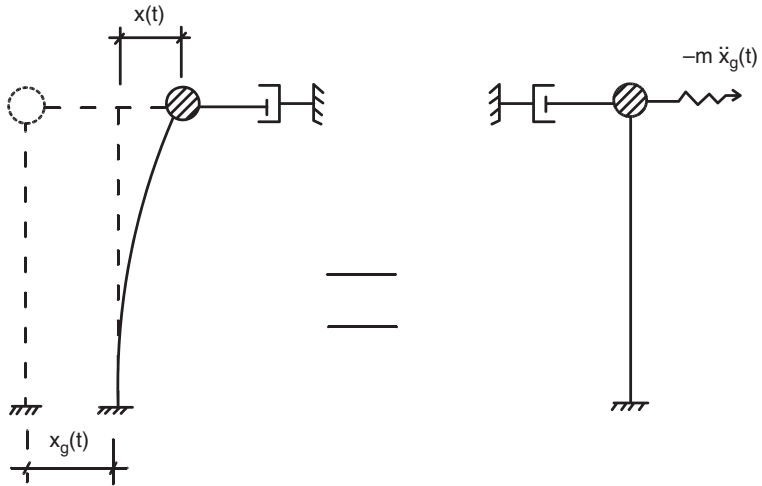
Fig. 4.2   SDOF under earthquake.
AutoCAD.

4.3.2   SDOF under earthquake

In the event of an earthquake, the external dynamic forces working on the structures result from the ground motion. Figure 4.2 shows this for a SDOF under the earthquake load.

The governing equation of motion now becomes:

$$-m[\ddot{x}_{g}(t) + \ddot{x}(t)] - c\dot{x}(t) - kx(t) = 0 \tag{4.2}$$

Where:

$x_{g}(t)$ is ground displacement
$x(t)$ is relative displacement
$\ddot{x}_{g}(t)$ is ground acceleration
$\ddot{x}(t)$ is relative acceleration, of single mass respectively
$\dot{x}(t)$ is relative velocity
$\ddot{x}_{g}(t) + \ddot{x}(t)$ is absolute acceleration due to the earthquake.

Fig. 4.3   Record of Imperial Valley earthquake.
Data downloaded from http://peer.berkeley.edu/nga/.

Let:

$$\omega = \sqrt{k/m} \tag{4.3}$$

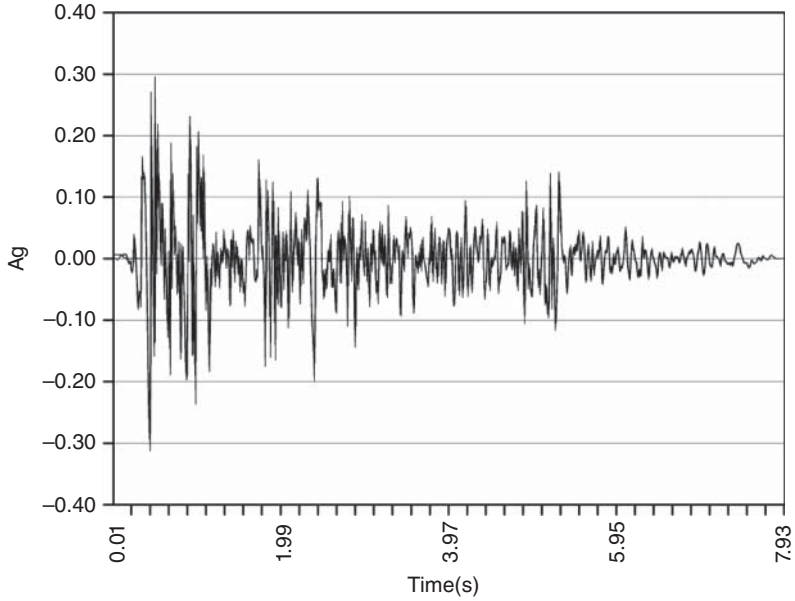Damping ratio:

$$\xi = \frac{c}{2m\omega} \tag{4.4}$$

Rearranging it, we get

$$\ddot{x}(t) + 2\xi\omega\dot{x}(t) + \omega x(t) = -\ddot{x}_g(t) \tag{4.5}$$

This is the governing equation of motion for the SDOF (single-degree-of-freedom structure) under earthquake loading.

Figure 4.3 is the ground acceleration q record of an Imperial Valley earthquake; it can be seen that, the ground acceleration $\ddot{x}_g(t)$ is a random distribution, therefore it is quite hard to solve the equation of motion using the normal method.

Using Duhamel's integral approach, the earthquake record can be treated as a sequence of short impulses, and the time-varying responses to each impulse are added together to give the total response.

For the basic equation of motion for SDOF under earthquake loading

$$\ddot{x}(t) + 2\xi\omega\dot{x}(t) + \omega x(t) = -\ddot{x}_g(t) \tag{4.5}$$

$-m\ddot{x}_g(t)$ can be treated as impulse force, here m =1

Using Duhamel's integral approach to integrate against the time, we can get the relative velocity

$$x(t) = -\frac{1}{\omega_D} \int_0^t \ddot{x}_g(\tau) e^{-\xi\omega(t-\tau)} \sin \omega_D(t-\tau) d\tau \tag{4.6}$$

Where:

ω_D is damped natural circular frequency

Therefore, the maximum displacement S_d for a SDOF system can be given as:

$$S_d = \left| -\frac{1}{\omega_D} \int_0^t \ddot{x}_g(\tau) e^{-\xi_i\omega_i(t-\tau)} \sin \omega_{Di}(t-\tau) d\tau \right|_{max} \tag{4.7}$$

Rearranging Equation 4.2, we get

$$-m[\ddot{x}_g(t) + \ddot{x}(t)] = c\dot{x}(t) + kx(t) \tag{4.8}$$

As $c\dot{x}(t)$ is much smaller than $kx(t)$, it can be ignored.

Therefore:

$$-m[\ddot{x}_g(t) + \ddot{x}(t)] \approx kx(t) \tag{4.9}$$

we can get

$$\ddot{x}_g(t) + \ddot{x}(t) = \frac{k}{m}x(t) = -\omega^2 x(t) \tag{4.10}$$

Inertial force

$$F(t) = -m[\ddot{x}_g(t) + \ddot{x}(t)] \tag{4.11}$$

So the total inertial force:

$$F(t) = kx(t) = m\omega^2 x(t) \tag{4.12}$$

$$x(t) = -\frac{1}{\omega_D} \int_0^t \ddot{x}_g(\tau) e^{-\xi\omega(t-\tau)} \sin \omega_D(t-\tau) d\tau \tag{4.13}$$

As $\omega_D \approx \omega$

Therefore, the inertial force can be given as:

$$F(t) = -m\omega \int_0^t \ddot{x}_g(\tau) e^{-\xi\omega(t-\tau)} \sin \omega(t-\tau) d\tau \tag{4.14}$$

The maximum seismic inertial force will be:

$$F(t)_{max} = -m\omega \left| \int_0^t \ddot{x}_g(\tau) e^{-\xi\omega(t-\tau)} \sin \omega(t-\tau) d\tau \right|_{max} \tag{4.15}$$

From Equation 4.11 and 4.14, we can get absolute acceleration:

$$\ddot{x}_g(t) + \ddot{x}(t) = -\omega \left| \int_0^t \ddot{x}_g(\tau) e^{-\xi\omega(t-\tau)} \sin\omega(t-\tau)d\tau \right|_{max} \tag{4.16}$$

Therefore, the maximum absolute acceleration:

$$S_a = |\ddot{x}_g(t) + \ddot{x}(t)|_{max} = \omega \left| \int_0^t \ddot{x}_g(\tau) e^{-\xi\omega(t-\tau)} \sin\omega(t-\tau)d\tau \right|_{max} \tag{4.17}$$

Similarly, we can also get the maximum velocity. These maximum values are to be used to construct the response spectrum, which we can use to design the structure under earthquake load. This will be introduced in Section 4.3.4.

4.3.3 MDOF under earthquake

It can be seen that not all structures (for an example, a multi-storey building) can be appropriately modelled as a SDOF system; a more complex multi-degree-of-freedom (MDOF) system is essential. This may respond in as many shapes as its degrees of freedom. Each mass in a MDOF system is generally subjected to inertia force, damping force, elastic force and external dynamic forces, as has been demonstrated in SDOF systems. Therefore, for dynamic equilibrium of any mass 'm', under earthquake loading, the governing equation of motion is:

$$[m]\{\ddot{x}\} + [c]\{\dot{x}\} + [k]\{x\} = -\ddot{x}_g(t)[m]\{I\} \tag{4.18}$$

Where:
$[m]$ is mass matrix
$[k]$ is the stiffness matrix
$[c]$ is the damping matrix
$\{I\}$ is unit vector
$\ddot{x}_g(t)$ is the ground acceleration.

The maximum response of a MDOF to seismic loading can be determined in a manner analogous to that used for SDOF; due to orthogonality, a MDOF system expressed in normal coordinates becomes uncoupled. Hence, the response of the system in any mode i (i =1, 2, 3, …) is equivalent to that of an independent SDOF system (Tedesco et al., 1999), for example the contribution of the ith mode to the total displacement of MDOF system is

$$\{x(t)\}_i = -\frac{\{\Phi\}_i \Gamma_i}{\omega_{Di}} \int_0^t \ddot{x}_g(\tau) e^{-\xi_i\omega_i(t-\tau)} \sin\omega_{Di}(t-\tau)d\tau \tag{4.19}$$

Where:
$\{\Phi\}_i$ is the modal vector of ith mode
Γ_i is the earthquake participation factor for ith mode.

The maximum response will depend on the integral's maximum value in Equation 4.19. Therefore, acceleration, velocity and displacement can be worked out

accordingly. As an example, the maximum value of the displacement in ith mode can be worked out as:

$$|\{x(t)\}_i|_{\max} = |\{\Phi\}_i \Gamma_i|(S_d)_i \tag{4.20}$$

Where:
S_d is the maximum displacement for the corresponding SDOF system.

After these maximum values have been worked out, the summation of these values for all the modes will provide the maximum value of the total response of the structure. However, the chance that all the modes achieve the maximum value at the same time is small, so more reasonable combination methods, such as square root of the sum of squares (SRSS) or complete quadratic combination (CQC), are widely used and are also available in commercial packages such as SAP2000, ETABS, Robot and Staad.Pro.

In square root of the sum of squares, the total response x is given by:

$$\{x(t)\}\max = \sqrt{\sum_{i=1}^{m} x(t)_{i\max}^2} \tag{4.21}$$

Where:
$\{x(t)\}\max$ is the maximum value for the total response
$x(t)_{i\max}$ is the peak value of response for the ith mode

The SRSS is accurate for a 2-dimensional structural system, for a 3-dimensional structure, the complete quadratic combination rule (CQC) proves to be more accurate. It is a generalisation of the SRSS rule and is applicable to a wider class of structures as it overcomes the limitation of the SRSS rule. It is specifically used for structures having closely spaced frequencies. The total response x is given by:

$$\{x(t)\}\max = \sqrt{\sum_{i=1}^{m} x(t)_{i\max}^2 + \sum_{i=1}^{m}\sum_{j=1}^{m} \rho_{ij} x(t)_{i\max} x(t)_{j\max}} \tag{4.22}$$

Where:
$\{x(t)\}$ max   is the maximum value for the total response
$x(t)_{i\max}$ is the peak value of response for the ith mode
$x(t)_{j\max}$ is the peak value of response for the jth mode

The second term in the above formula is valid for $i \neq j$. Thus, the second term includes the effect of correlation between the modal peak responses through the correlation coefficient terms ρ_{ij} (Chopra, 2006).

4.3.4 *Response spectrum*

As mentioned in Sections 4.4.2 and 4.4.3, using Duhamel's integral approach we can easily obtain the maximum acceleration, velocity and displacement for a certain earthquake. However, in earthquake design we are only concerned with the maximum dynamic response.

In order to summarise the response of all possible SDOF structures due to a particular component of ground motion, a response spectrum is used. It is used to plot the peak value of a response quantity (e.g. displacement, velocity or acceleration) as a function of the natural vibration frequency, f, ω, or the natural period T (Tedesco et al., 1999). This plot is called the response spectrum for that quantity.

For MDOF, using the response spectrum, the maximum response of each mode can be obtained. Therefore, the total response of the whole structure can be obtained using either Equation 4.21 or 4.22.

4.3.5 Modal analysis

Modal analysis is important to response spectrum analysis, as the maximum response of the structure is based on the superposition of the response of each mode. Modal analysis is used to determine the fundamental vibration mode shape and the corresponding frequency of the structures. Most of the structural computer software such as SAP2000, ETABS, Staad.Pro or Robot can perform the modal analysis.

According to the modal shapes, the horizontal elastic response spectrum of each element of the structure can be determined in programs such as SAP2000. In theory, one should include all the possible modes in the analysis; however, in reality, to reduce the computation time it is essential to choose the minimum but sufficient modes during the analysis. From engineering practice, the author would recommend at least 100 modes to be selected for the modal analysis either in SAP2000 or ETABS, and check the sum effective model masses is at least 90% of the total structural mass (how to check this will be shown later). This is to follow what is required by Eurocode 8 (2004) that, 'in the analysis, one should either include sufficient modes that the sum effective model masses is at least 90% of the total structural mass, or to include all modes with an effective modal mass greater than 5% of the total mass'.

4.3.6 Response spectrum from Eurocode 8

For designers, design codes such as Eurocode 8 (2004) give a detailed definition of the response spectrum of an earthquake. It is defined by enveloping and smoothing spectra corresponding to different earthquake time histories. The maximum acceleration response of the structures is determined over the expected earthquake ground motion. It is measured as ground acceleration according to the gravity acceleration (g). The elastic response spectrum by Eurocode 8 (2004) is shown in Figure 4.4. There are two types of elastic spectra curves that are related to the magnitude of an earthquake and the soil type lying under the structure's foundations. Eurocode 8 (2004) also gives the formula to construct a design spectrum with the use of the q factor, this will be introduced in the next section, please refer to Eurocode 8 (2004) for further details. Figure 4.4 is the horizontal elastic response spectrum from Eurocode 8 (2004).

These response spectrums can be directly defined in programs such as SAP2000, ETABS, Staad.Pro and Robot, and will be demonstrated in Section 4.4.
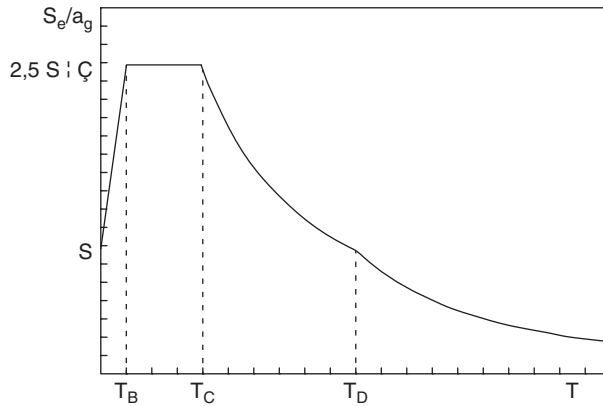
Fig. 4.4   Horizontal elastic response spectrum.
Reproduced according to Eurocode 8: Design of Structures for Earthquake Resistance –
Part 1: General Rules, Seismic Actions and Rules for Buildings.

4.3.7   Ductility and modified response spectrum

Designing a structure to remain elastic during large earthquakes is not economical; it is reasonable to make use of the ductility of the structure to dissipate the input energy from earthquake, and therefore reduce the structural member size. In order to design the structure based on the ductility, it is essential to ensure that the structure fails in a ductile manner, rather than a brittle failure which means the ductility supply exceeds the maximum demand: this is the so-called capacity design approach. Normally, it can be achieved in the design as follows:

• Design a weak beam, but a strong column.
• Strong beam to column connections, to make sure the structure fails in the structural member rather than the connection. For a concrete structure, this can be achieved by adequate confinement to concrete using closely spaced steel hoops. For steel structures this can be achieved by reducing the flange of the beam near the connection location.
• Avoid shear failure as it is normally brittle.
• Avoid large irregularities.

q-factor

Ductility helps the energy absorption of the frame elements during an earthquake. It can be expressed using the q-factor in European codes and a similar factor can be found in US design codes, and defines the dissipative behaviour of the structure. Ductile element-members can absorb the earthquake loadings and therefore have the capacity to resist more earthquake load.

In Eurocode 8 (2004), factor q is an approximation of the ratio of earthquake loadings that the structure would experience if its response was perfectly elastic with 5% viscous damping, to the earthquake loading that is taken into account during the

Table 4.1 **Limiting values of q factors for steel structures.**

Design concept	Structural ductility class	Range of the reference values of the behaviour factor q
Concept a) Low dissipative structural behaviour	DCL(Low)	≦1.5-2
Concept b) Dissipative structural behaviour	DCM(Medium)	≦4 also limited by the values of Table 6.2
	DCH(High)	only limited by the values of Table 6.2

Eurocode 8: Design of Structures for Earthquake Resistance - Part 1: General Rules, Seismic Actions and Rules for Buildings, page 137.[1]

design. This factor is vital because it helps to ensure that the structure will have a satisfactory response during an earthquake. The behaviour factor q also account for the influence of the viscous damping being different from 5%. Eurocode 8 (2004) defines the value of q in different Ductility Classes as shown Table 4.1.

Ductility modified response spectrum

To make use of the ductility, both Eurocode 8 (2004) and ASCE/SEI 7-10(2010) used an approximate analysis method that applied a so-called ductility modified response spectrum, which is called a design spectrum. Over most of the period range the spectral acceleration S_d (and also the design force) are a factor of q times lower than the value S_e (elastic spectrum) for the equivalent elastic system. In commercial programs such as SAP2000 and ETABS, the reader can define this q factor in the response spectrum function, which will be introduced in the next section.

4.4   Modelling example of the response spectrum analysis using SAP20001

In this section, a modelling example of the response spectrum of a multi-storey building using SAP2000 will be introduced. This structure will be hypothetically designed in a region with high seismic risk.

Step 1: Model set up

- Click on *New*, a window will pop up as shown in Figure 4.5. Input data of the building. Choose the correct units, *KN*, *m*, *C*, then select the template of *3D Frames* for this example.

---

[1] Permission to reproduce and derive extracts from British and ISO standards is granted by BSI. British Standards can be obtained in PDF or hard copy formats from the BSI online shop: www.bsigroup.com/Shop or by contacting BSI Customer Services for hardcopies only: Tel: +44 (0)20 8996 9001, Email: cservices @bsigroup.com.
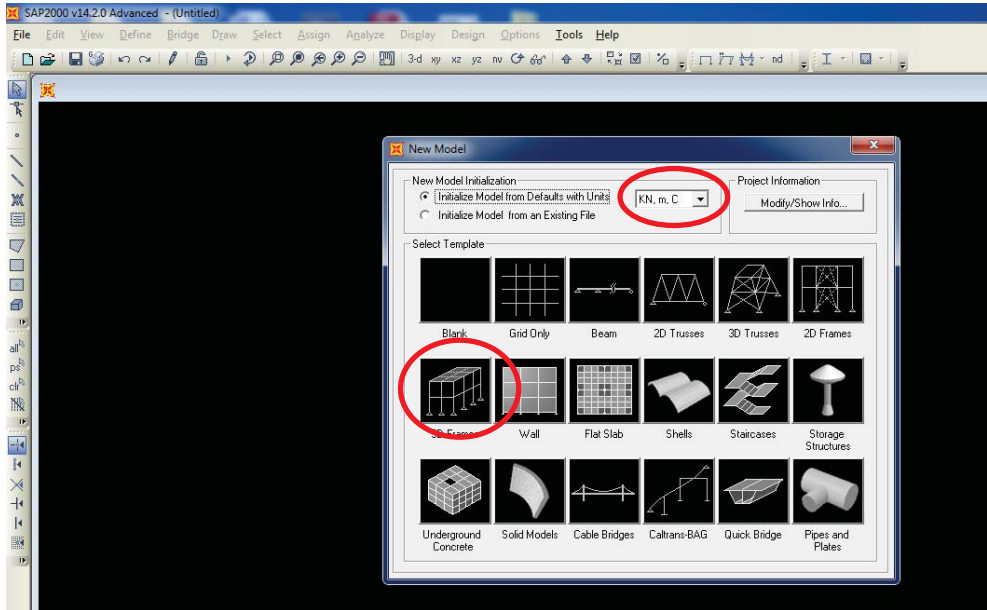
Fig. 4.5   Input data of the building.
SAP2000 screenshot reprinted with permission of CSI.

Step 2: Building configuration

The geometry of the building (height, area etc.) will be defined, as well as the frame type. Moreover, the section properties of the structure (beams, columns, slabs) will be selected. The material properties, such as the concrete and steel, are also defined. The geometry can also be imported from AutoCAD or Rhino, which has been introduced in earlier sections. Here, we just define a regular frame as shown in Figure 4.6.

Step 3: Define response spectrum function

In this step, the response spectrum function and the parameters will be defined.

- Click *Define*, then *Functions*, choose *Response Spectrum*, as shown in Figure 4.7. A window will pop up as shown in Figure 4.8.
- In the *Choose Function Type Option*, choose Eurocode 8 (2004), click *Add New Function*.
  A window will pop up as shown in Figure 4.9.
- Define the required parameters as it demonstrates.

Parameters such as the design ground acceleration can be checked in the ground acceleration maps from design codes such as Eurocode 8 (2004) and ASCE/SEI 7-10, (2010). Other design parameters such as spectrum type, ground type and behaviour
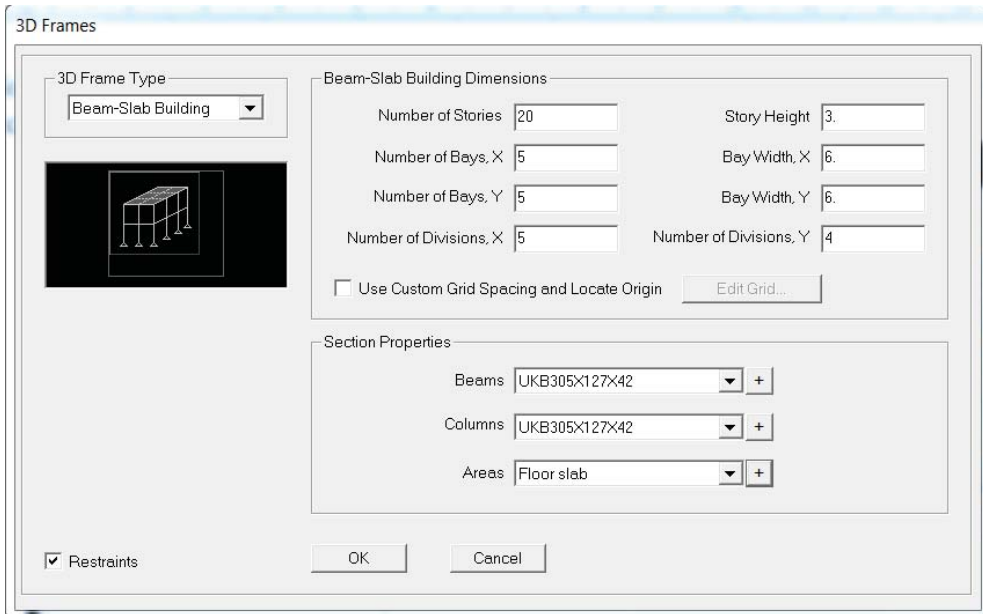
Fig. 4.6 Define the 3D frame.
SAP2000 screenshot reprinted with permission of CSI.
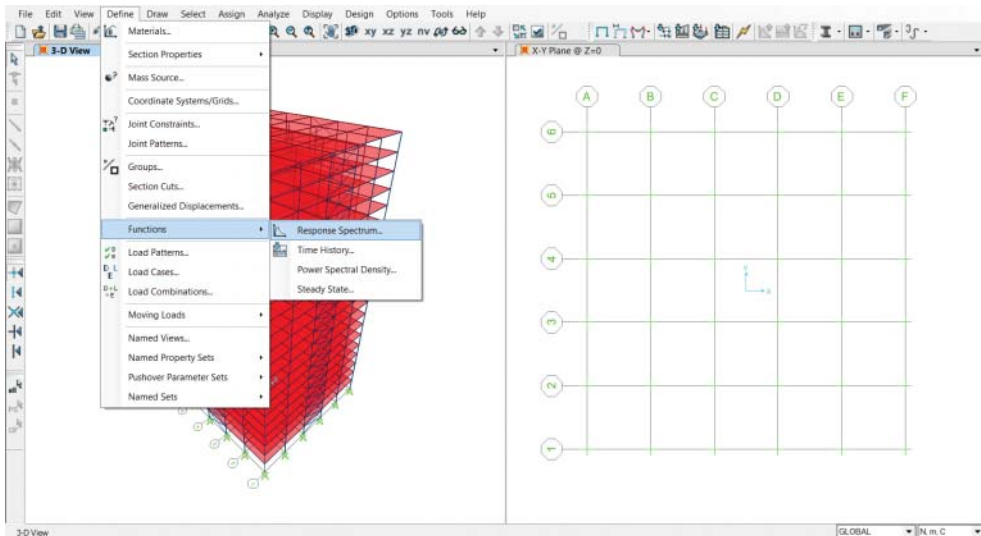
Fig. 4.7 Choosing the response spectrum function.
SAP2000 screenshot reprinted with permission of CSI.
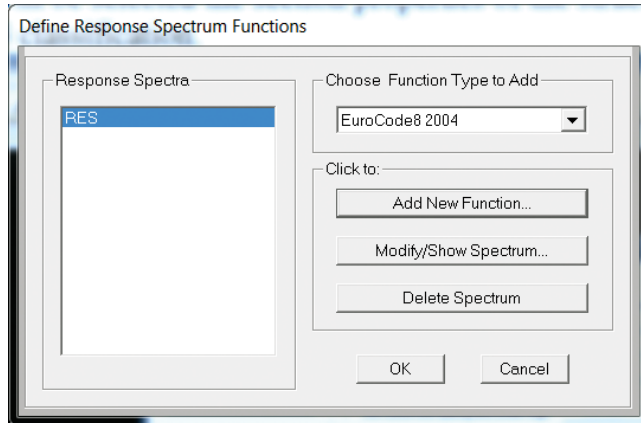
Fig. 4.8   Defining the response spectrum function.
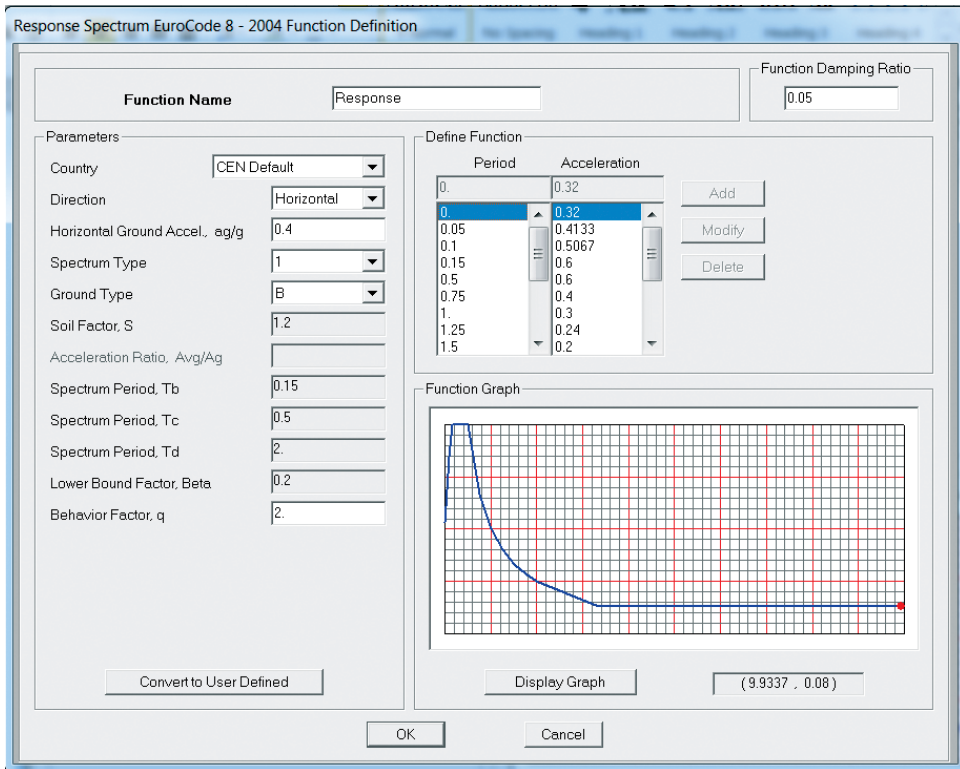**SAP2000 screenshot reprinted with permission of CSI.**

Fig. 4.9   Definition of horizontal response spectrum.
SAP2000 screenshot reprinted with permission of CSI.

factor can also be determined according to the Eurocode 8 (2004) and ASCE/SEI 7-10, (2010).

- In the *Direction Option*, choose horizontal or vertical ground acceleration direction. Similarly you can also define the vertical response spectrum.

Step 4: Define load pattern

- Click *Define*, *Load Patterns*, choose *Add New Load Pattern* and define the response spectrum load pattern as shown in Figure 4.10.

Step 5: Load case data

- Click on *Define Load Case – Add New Load Case*, as shown in Figure 4.11. A new load case window will pop up, as shown in Figure 4.12.
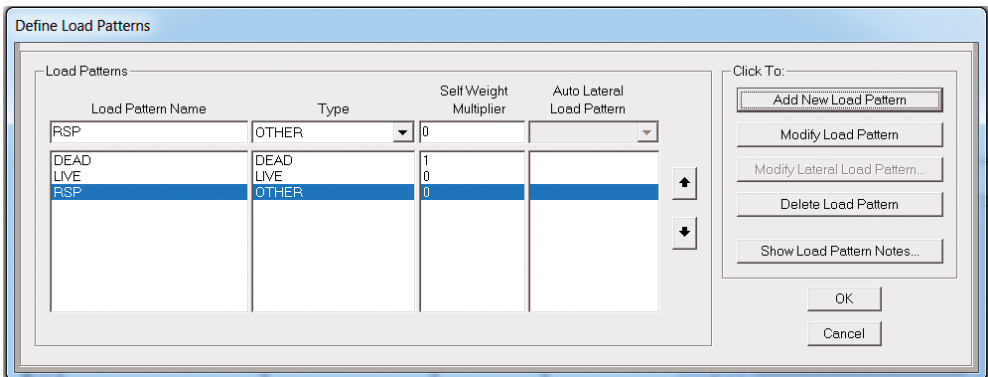
Fig. 4.10   Response spectrum load pattern.
SAP2000 screenshot reprinted with permission of CSI.
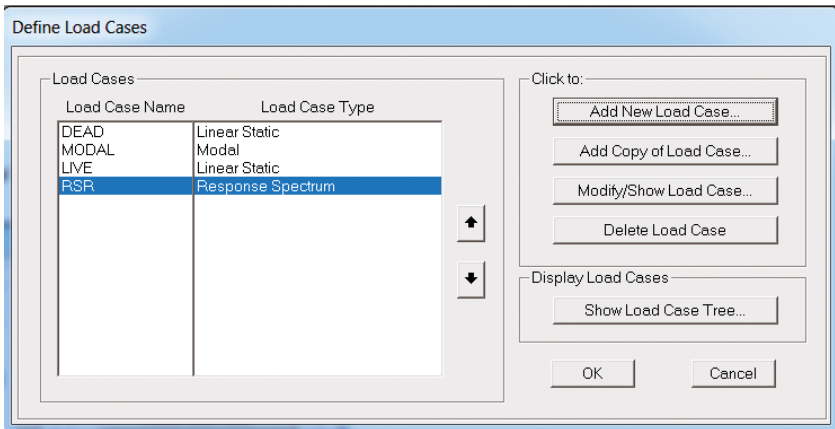
Fig. 4.11   Defining load cases.
SAP2000 screenshot reprinted with permission of CSI.
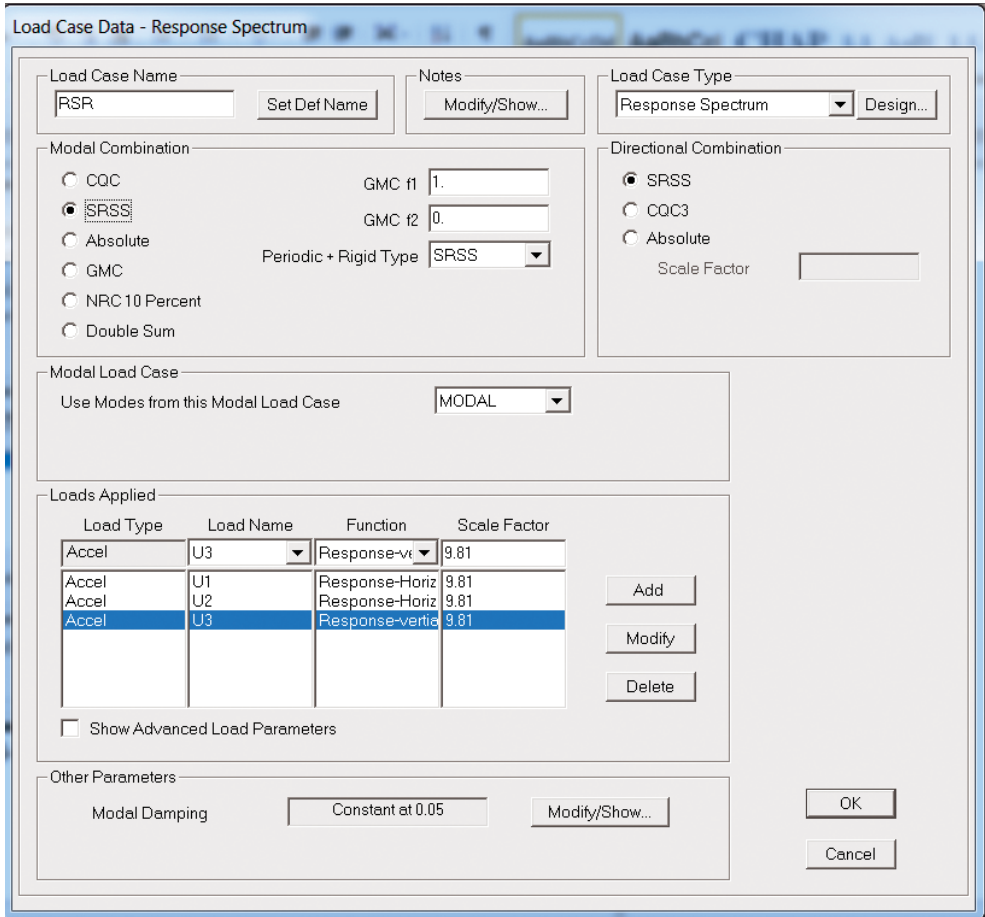
Fig. 4.12   Defining load case parameters for response spectrum.
SAP2000 screenshot reprinted with permission of CSI.

- In the *Load Case Type* choose *Response Spectrum*.
- In the modal combination choose *CQC* or *SRSS*; these two methods were introduced in Section 4.3.

As shown in Eurocode 8 (2004), horizontal seismic motion is a bi-directional phenomenon and thus the building structure should be able to resist horizontal actions in any direction.

- In the *Directional Combination Option*, you can define how to combine the earthquake in U1, U2 and direction (it stands for x,y direction). We choose *SRSS* here.
- In the *Loads Applied* section, define the *Acceleration Function* in U1, U2 and U3(x,y,z) direction, choose the horizontal and vertical response defined in Step 3.
- In the *Scale Factor* option, choose *9.81* which is the gravity acceleration, this is because in the response spectrum defined in Step 3, the unit of acceleration is in term of $g = 9.81$ m/s^2 (acceleration of gravity).
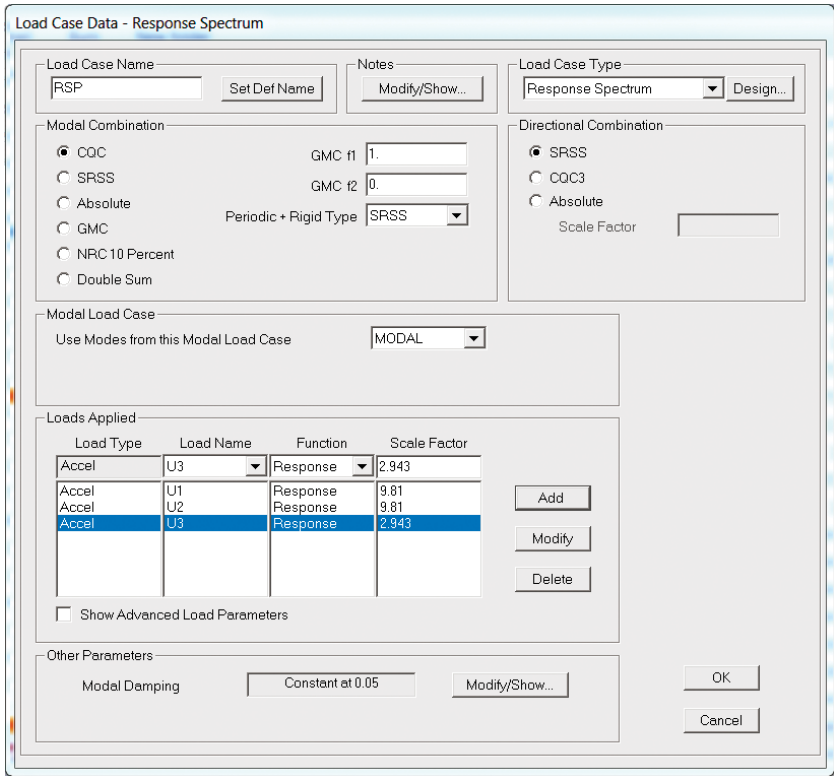
Fig. 4.13   Defining load cases for response spectrum.
SAP2000 screenshot reprinted with permission of CSI.

- If the vertical response is not defined separately, it is important to note that acceleration in the U3 direction should be further multiplied by a scale factor of 0.3, that is 9.81*0.3=2.943 in the U3 direction. This is following Eurocode 8 (2004); the vertical acceleration can be approximate as 30% of horizontal acceleration, see Figure 4.13.

It is also important to define the modal analysis, as explained earlier; SAP2000 will perform a modal response spectrum analysis, one should include sufficient modes to make sure the sum effective model masses are at least 90% of the total structural mass. Therefore, at least 100 modes should be selected for the modal, as shown in Figure 4.14.

Step 6: Define load combination

Following Eurocode 8 (2004), the load combination can be defined as follows in SAP2000:

$$q = G_{\mathrm{K}} + 0.3 \times Q_{\mathrm{K}} + A_{\mathrm{Ed}}$$
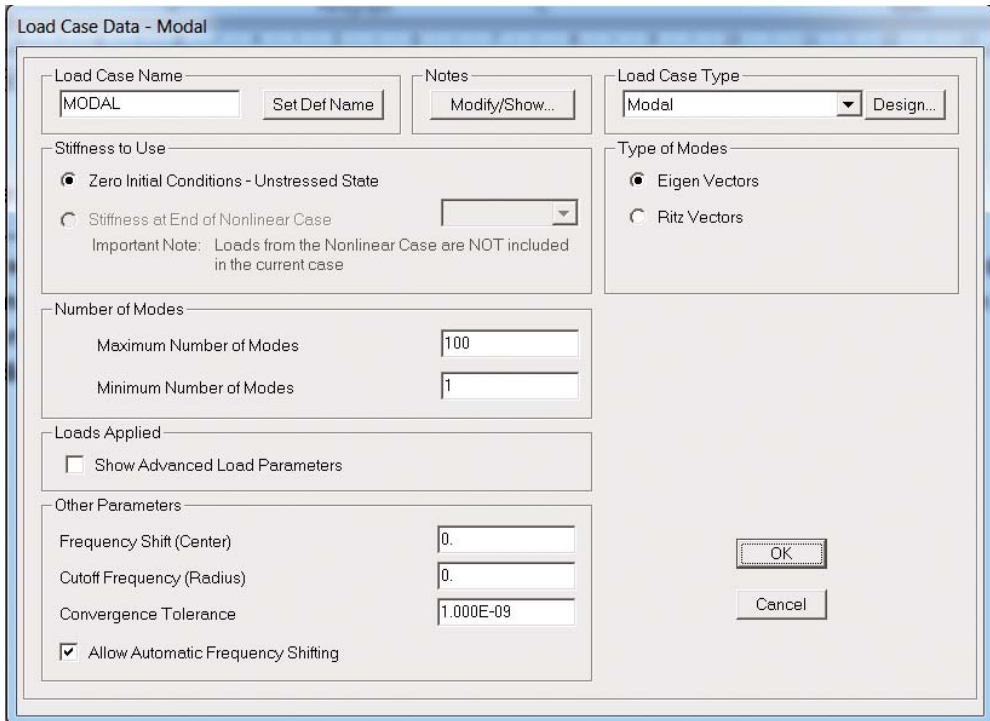
Fig. 4.14   Defining the modal case.
SAP2000 screenshot reprinted with permission of CSI.

Where:

G_K is the dead load
Q_K is the live load
A_{Ed} is the design value of the seismic action.

- Choose *Define* then *Load Combination*.
- A window will pop up as shown in Figure 4.15, *Add New Combination*.
- A window will pop up as shown in Figure 4.16, then set up the parameters as shown.

Step 7: Run analysis

- Select *Analyze – Run Analysis*, the below window will pop up (Figure 4.17), and then choose *Run Now*.

Step 8: Interpretation of the analysis results

As mentioned in the previous section, as required by Eurocode 8 (2004), after the analysis, it is important to check that the participation of the mass of the structure has
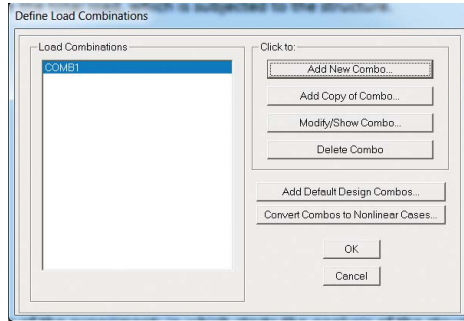
Fig. 4.15   Defining the load combination.
SAP2000 screenshot reprinted with permission of CSI.
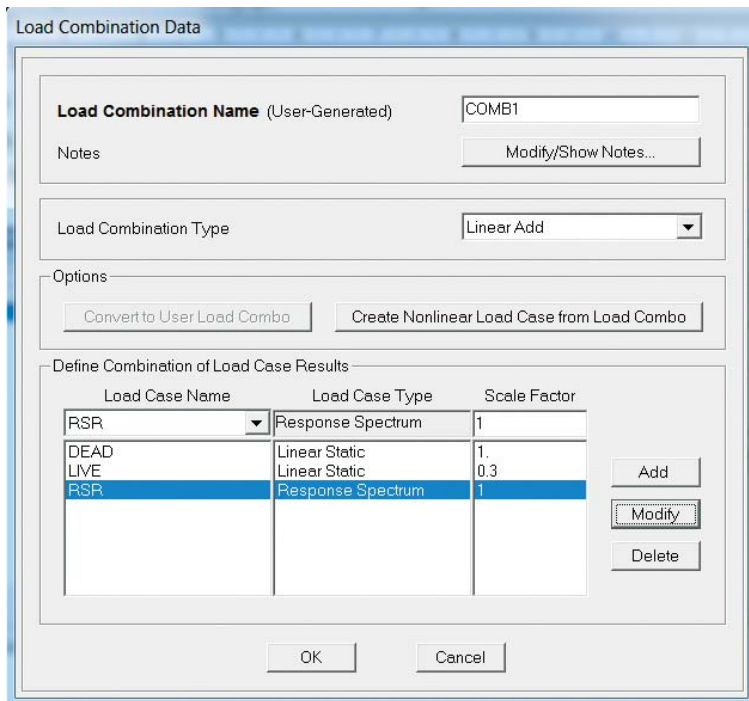
Fig. 4.16   Defining the load combination (2).
SAP2000 screenshot reprinted with permission of CSI.

achieved more than 90% of the overall mass of the building.

- Click *Display* then select *Show Table*.
- A window will pop up as shown in Figure 4.18, then select *Joint Output*.
- In the *Joint Mass* tab shown in Figure 4.19, choose *Modal Load Participation Ratios*, check the ratio percentage is at least 90% in both x and y directions.
- Similarly we can obtain the internal forces and displacements. The internal forces and displacements can be further exported into Excel files, as shown in Figure 4.20.
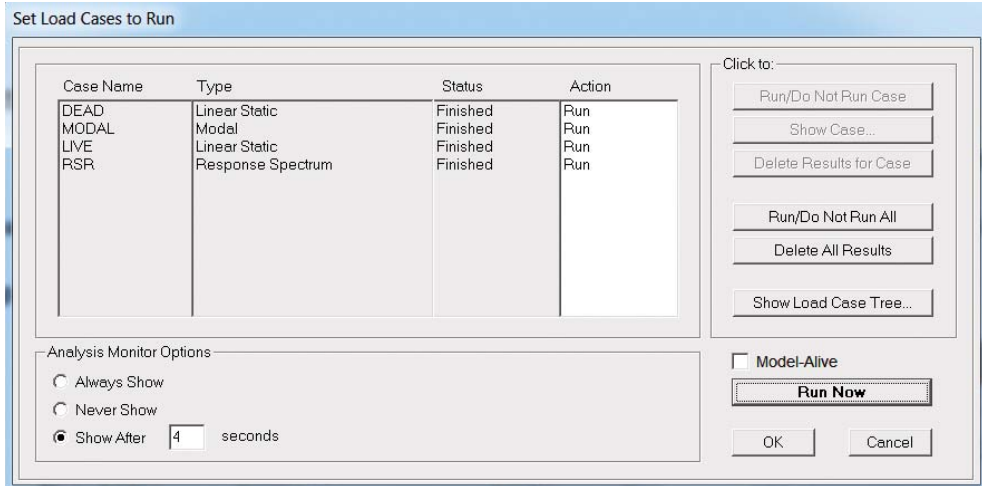
Fig. 4.17   Running the analysis.
SAP2000 screenshot reprinted with permission of CSI.
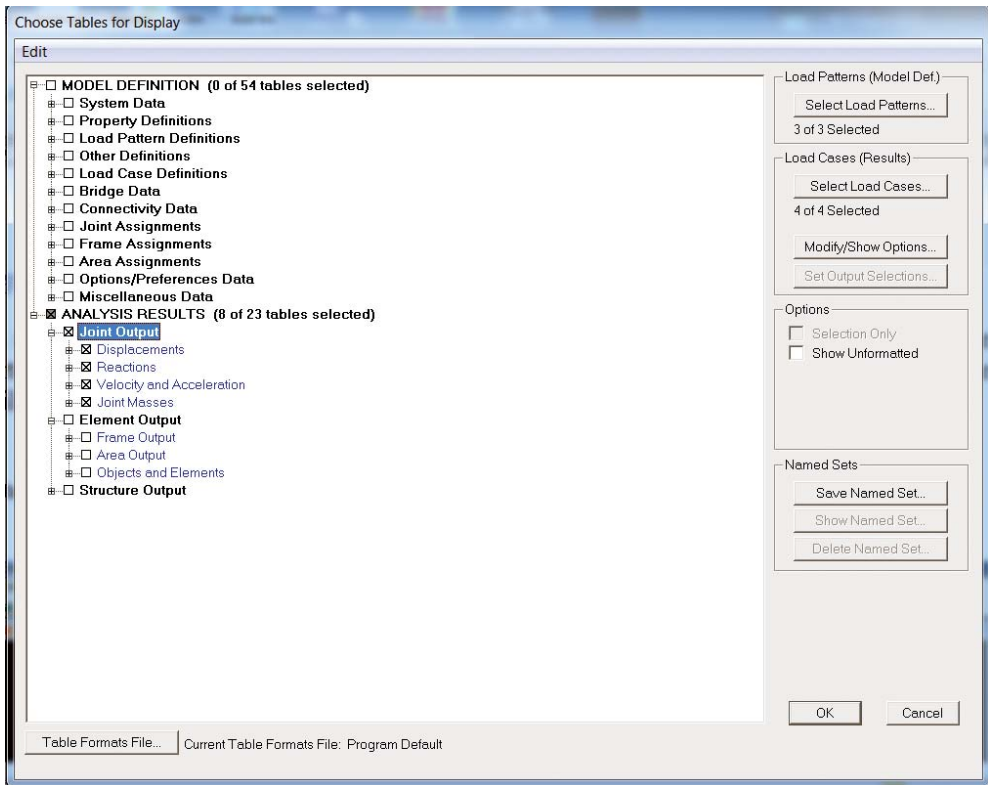
Fig. 4.18   Result interpretation.
SAP2000 screenshot reprinted with permission of CSI.

Fig. 4.19 Modal load participation ratio check.
SAP2000 screenshot reprinted with permission of CSI.

Fig. 4.20 Internal forces and displacements.
SAP2000 screenshot reprinted with permission of CSI.

4.5   Time history analysis and modelling example using SAP2000

Another widely used dynamic analysis method is time history analysis. It is used to analyse under a ground acceleration the time history with the frequency content that matches the design spectrum. The time history is specified as a series of data points at the time interval, such as 0.01 s, and the analysis is performed using a stepwise direct integration. As different time histories may provide different structural responses, Eurocode 8 (2004) requires that at least three analyses under different accelerograms must be performed. If at least seven different analyses are performed then the mean results may be used; otherwise the most onerous result should be used. As huge computational cost is involved in time history analysis, this method is more applicable to using a computer program. Most structural computer software, such as SAP2000, ETABS, Staad.Pro or Robot, can perform time history analysis.

4.5.1   Fundamentals of time history analysis

As discussed in Section 4.4, for a SDOF, the basic equation of motion is

$$m\ddot{x} + c\dot{x} + kx = F(t) \tag{4.1}$$

If we divide the applied force $F(t)$ into small segments with the time interval of Δt, if Δt is small enough, then, at any time $t=i$

$$m\ddot{x}_i + c\dot{x}_i + kx_i = F(t)_i \tag{4.23}$$

at any time $t=i+1$

$$m\ddot{x}_{i+1} + c\dot{x}_{i+1} + kx_{i+1} = F(t)_{i+1} \tag{4.24}$$

If we know the initial conditions such as initial displacement x_0 and initial velocity $\dot{x}_0$ we can apply successively with $i=0, 1, 2, \ldots$ then we can work out the response at all times $t=1, 2, 3, \ldots$ therefore the overall response of the structure can be obtained.

Stepping from i to $i+1$ can be done with approximate procedures such as wilson-θ and Newmark, the reader can refer to books such as Chopra (2006) or Tedesco et al. (1999) for further study.

4.5.2   Modelling example of time history analysis using SAP2000

Similar to response spectrum analysis, we can set up a multi-storey building in SAP2000.

Define the functions of time history

- Click *Define* then select *Functions*, then select *Time History* as shown in Figure 4.21.
- Download the time history of the ground acceleration. Define the accelerogram in SAP2000.
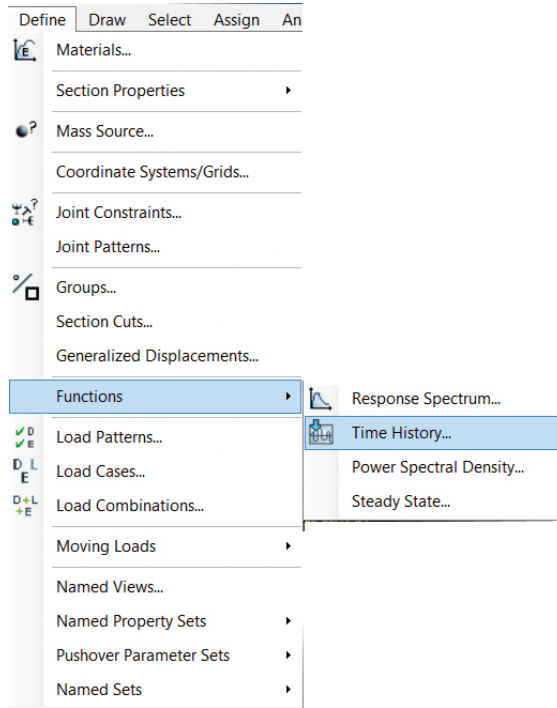
Fig. 4.21 Choosing the functions of time history.
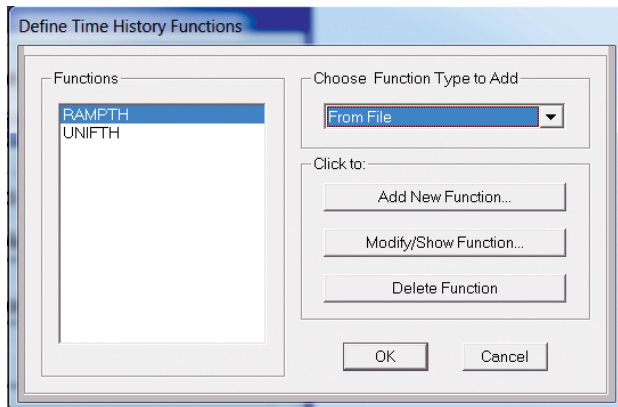**SAP2000 screenshot reprinted with permission of CSI.**

Fig. 4.22 Choosing the time history record file.
SAP2000 screenshot reprinted with permission of CSI.

- In the ribbon, click *Define*, *Load Case*. Define the *Time History* case.
- As shown in Figure 4.22, the accelerogram can be chosen from *Function Type – From File*, an option that can be selected from the drop-down list.
- A window will pop up as shown in Figure 4.23, choose the time history files from a local drive by clicking *Browse*, the time history files can be download from a website such as http://peer.berkeley.edu/nga.
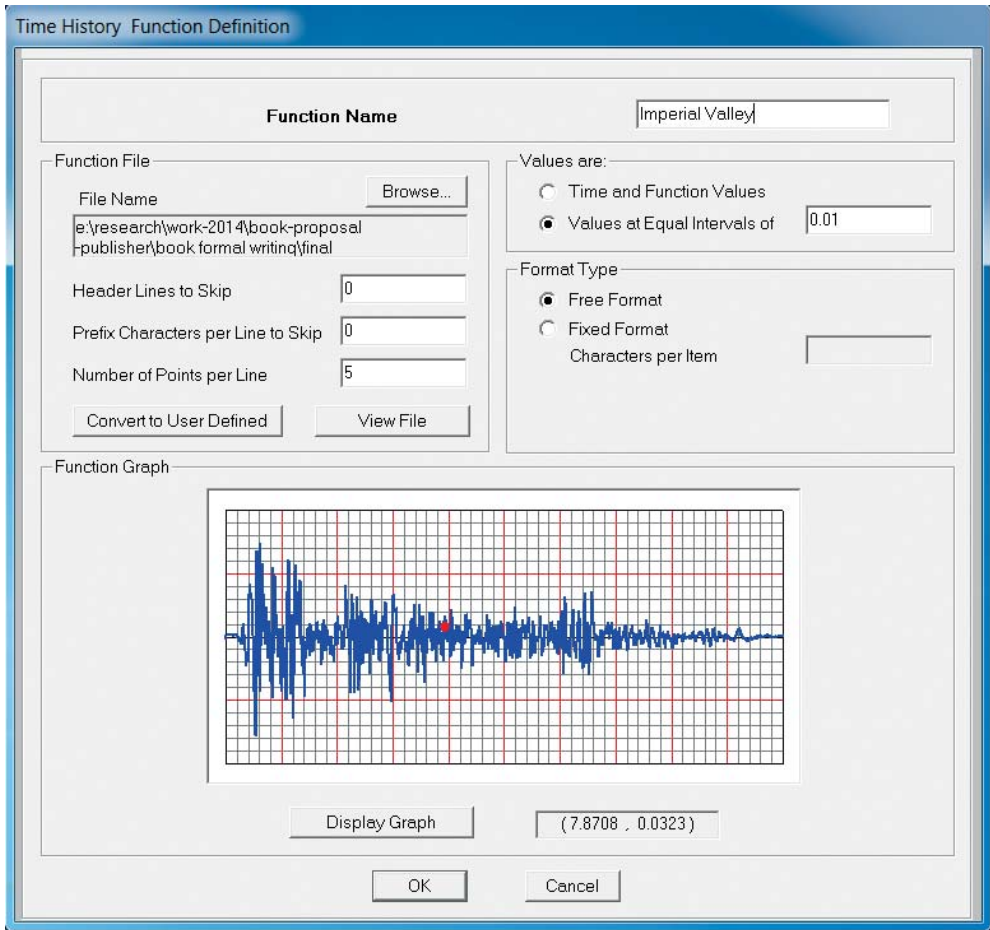
Fig. 4.23   Defining the time history function.
SAP2000 screenshot reprinted with permission of CSI.

- Give the name, for example 'Imperial Valley'.
- Choose the *Values at Equal Interval* 0.01.
- By clicking on *Display Graph* we can visually check the waveform.
- Go to *Define – Load Cases – Add New Load Case*.
- A new window will pop up as shown in Figure 4.24.
- If the records are given in units of g, the scale factor would be 9.81 m/s^2.
- The number of time steps should be given by the duration of the accelerogram divided by the sampling time. In our example: 40 s / 0.01 s = 4000 steps as shown in Figure 4.24.

Analysis results

The analysis result can be interpreted either through *Display – Show Table* as was introduced earlier, or following the below steps:
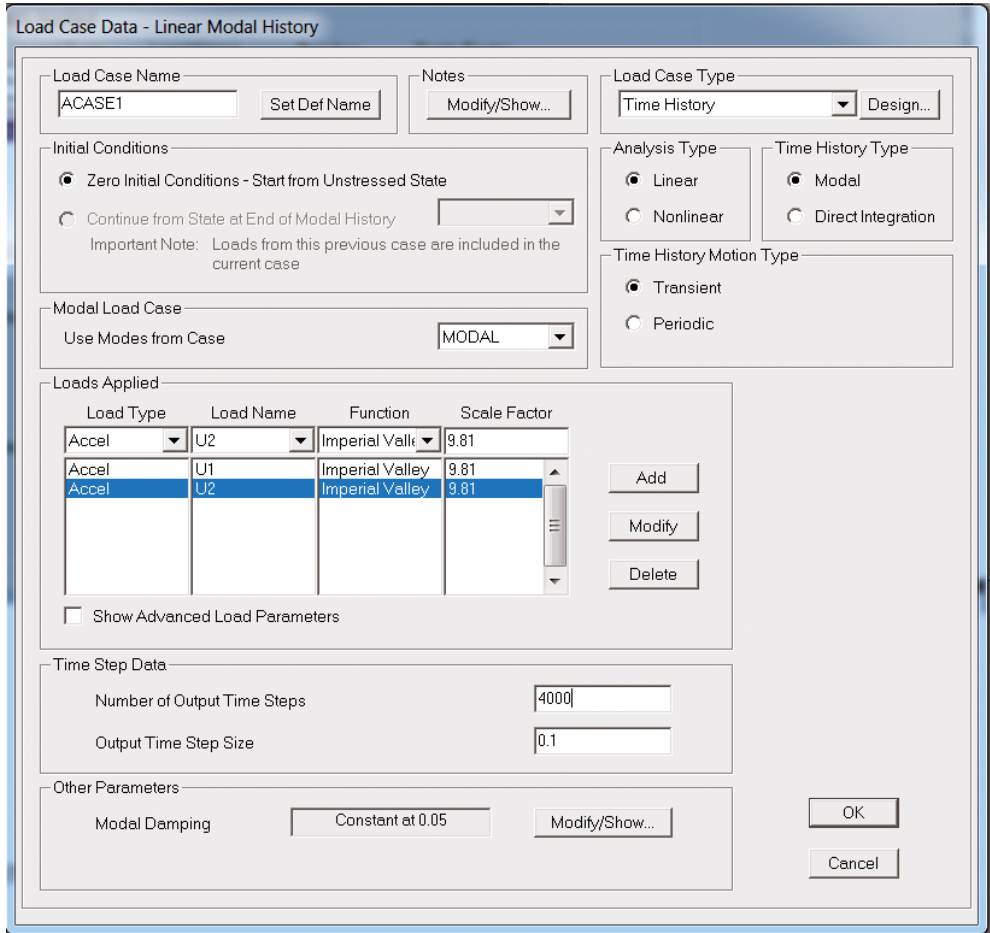
Fig. 4.24   Defining the parameters of time history.
SAP2000 screenshot reprinted with permission of CSI.

- Click *Display* – show *Plot Functions*.
- A window will pop up as shown in Figure 4.25.

You can display the time history of the output you want, such as the displacement of certain nodes or base shear in the plots. Here, we use base shear as an example to demonstrate how to plot the time histories.

- Click *Define Plot Functions* in Figure 4.25.
- A new window will pop up as shown in Figure 4.26, choose add *Base Functions*.
- Choose *Base Shear X* and *Base Shear Y*, as shown in Figure 4.27, then click *OK*.
- Then add the *Base Shear X* and *Base Shear Y* function as shown in Figure 4.28.
- The time history result of *Base Shear X* and *Base Shear Y* are shown in Figure 4.29.
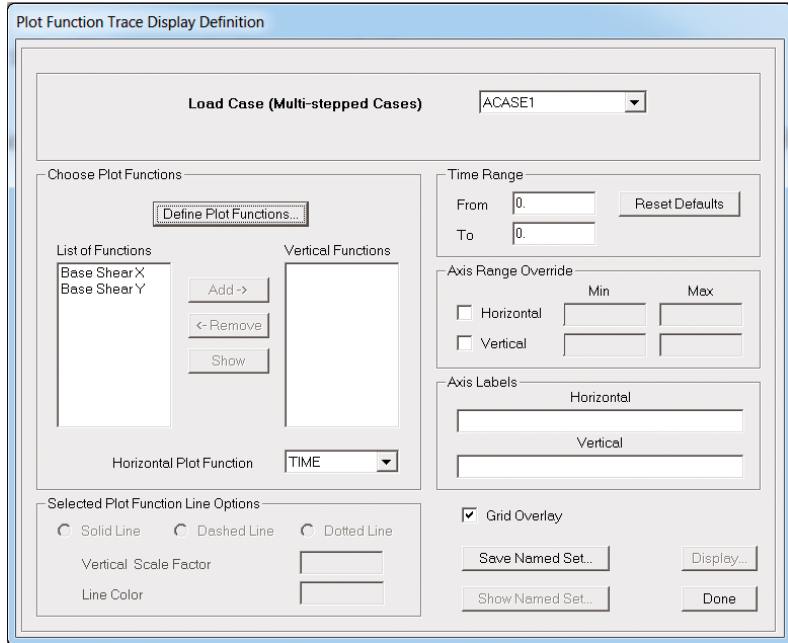
Fig. 4.25 Defining the time history plot for base shear.
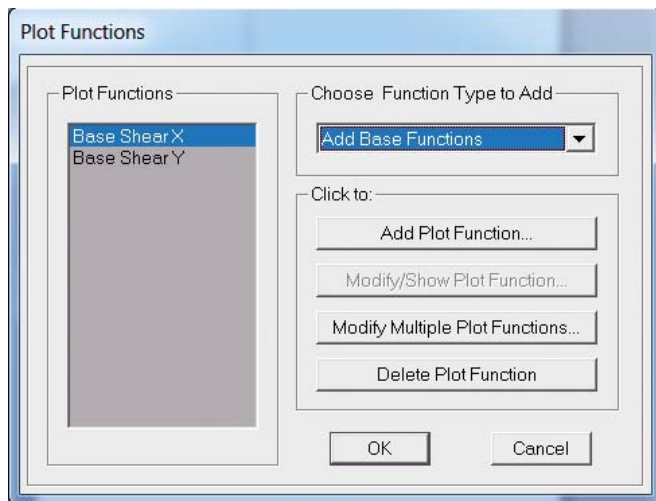SAP2000 screenshot reprinted with permission of CSI.

Fig. 4.26 Base shear function.
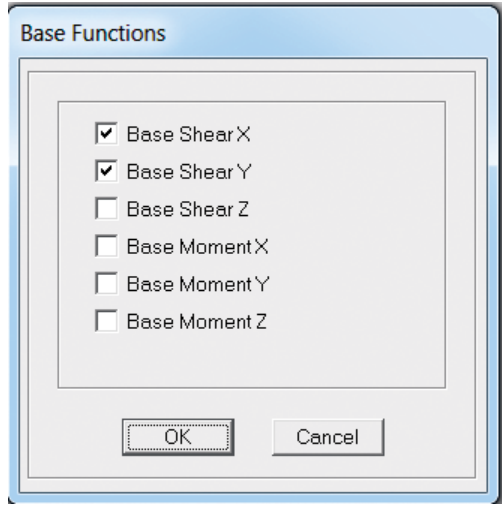SAP2000 screenshot reprinted with permission of CSI.

Fig. 4.27   Choosing *Base Shear X* and *Base Shear Y*.
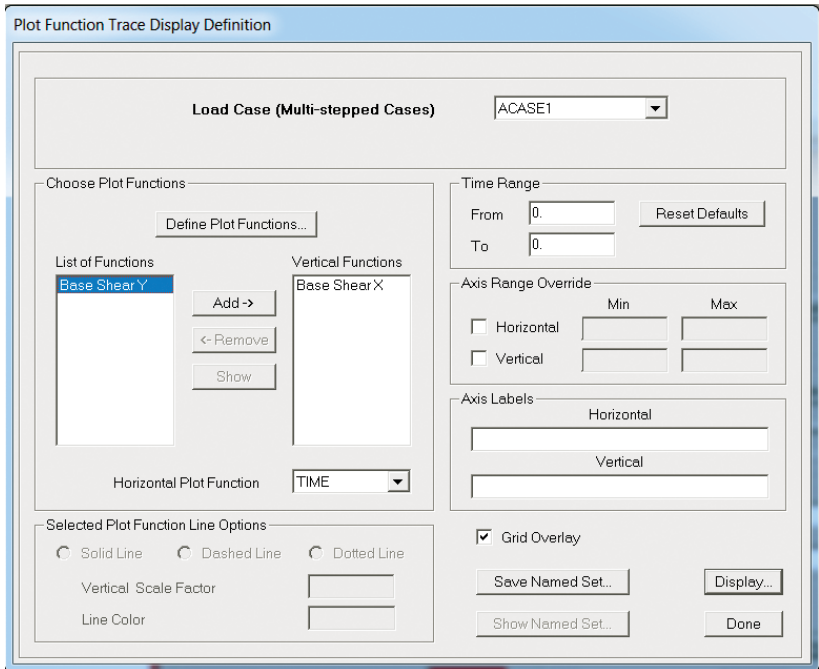**SAP2000 screenshot reprinted with permission of CSI.**

Fig. 4.28   Display the time history.
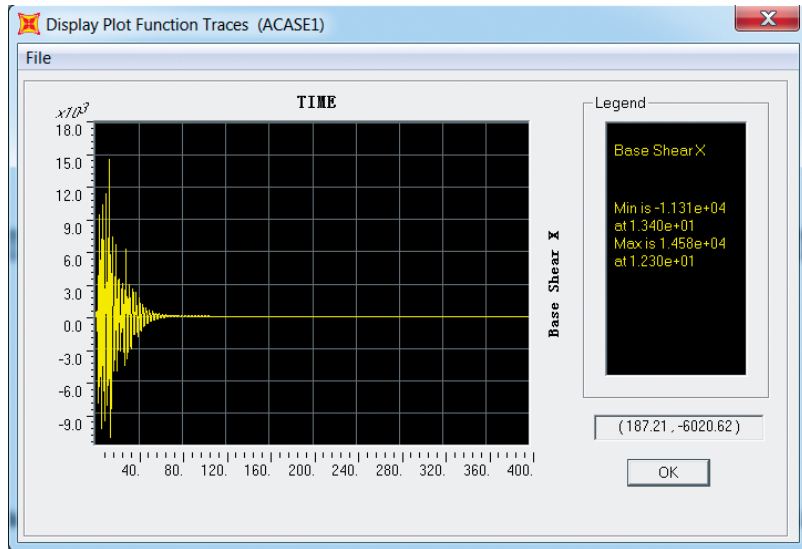**SAP2000 screenshot reprinted with permission of CSI.**

Fig. 4.29   Base shear time history.
SAP2000 screenshot reprinted with permission of CSI.

4.6   Push-over analysis and modelling example using SAP2000

Push-over analysis or so-called nonlinear static analysis is another important approach for earthquake analysis. In the past, push-over analysis was known as the nonlinear capacity estimating tool, hence it is also known as capacity analysis. Rather than performing a dynamic analysis, a static analysis is performed.

4.6.1   Introduction

In this analysis, the building is subjected to a monotonically increasing lateral load with a variant distribution until a certain target displacement is obtained. A nonlinear static analysis is performed at each step, until the building forms a collapse mechanism, which can be checked through the development of plastic hinge distribution in software such as SAP2000.

A 2D or 3D structure that includes a bilinear and trilinear deformation-load diagram of the lateral load resisting elements in the building is created first with a predefined pattern of lateral load is defined, which is distributed along the building height. The lateral force is increased until some elements such as beams or columns yield. The structural model is then modified against reduced stiffness of the yielding members and the lateral load is again increased until more members start yielding, the yielding can be monitored by the definition of the plastic hinge in software such as SAP2000. This process continues until a target displacement at the top of a building reaches certain levels of deformation.

The analysis results can be interpreted through a push-over curve, which is base shear against top displacement, and is also called a capacity curve. By equating the

capacity curve to the demand curve (such as an earthquake response spectrum), the peak displacement (so-called target displacement) under the design earthquake can be obtained.

Unlike the modified response spectrum method introduced in Section 4.3, which assumes a uniform ductility by a simple scale-down of a behaviour factor, the advantage of the push-over analysis is that it is based on an accurate estimation of the distributed ductility of the structure.

4.6.2   Modelling example of push-over analysis using SAP2000

Step 1: Set up a multi-storey builidng and initial analysis

- Similar to Section 4.5, a 3D model is first created in SAP2000, as shown in Figure 4.30, a multi-storey building is built with lateral resistance using MRF.
- Run linear analysis under *Dead*, *Live* and *Response Spectrum* analysis, you can choose *Auto-selection Section List* to let SAP2000 design the structural members.

Step 2: Plastic hinge definition

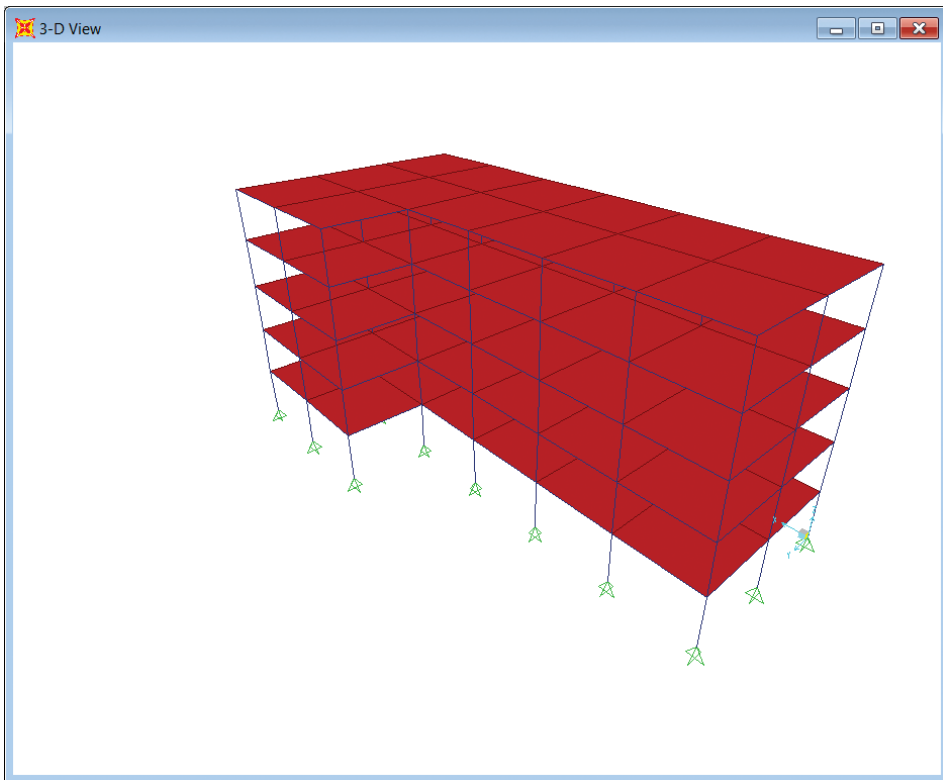From the *Define* menu choose *Hinge Properties* as shown in Figure 4.31.

Fig. 4.30   The 3D model.
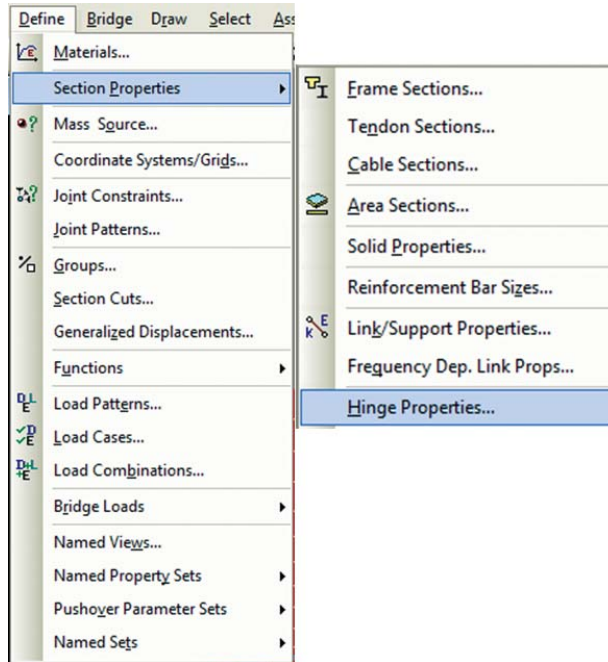**SAP2000 screenshot reprinted with permission of CSI.**

Fig. 4.31   Plastic hinge definition in SAP2000.
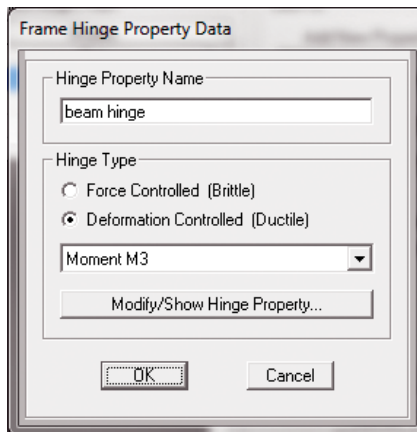**SAP2000 screenshot reprinted with permission of CSI.**

Fig. 4.32   Steel beam plastic hinge definition.
SAP2000 screenshot reprinted with permission of CSI.

There are different kinds of plastic hinges available in SAP2000, one for steel beams, one for steel columns, one for steel circular bracings and one for reinforced concrete shear walls. We are going to introduce the definition of the steel beam here; the steel beam plastic hinge is defined to be at the ends of the beams in terms of moment on the major axis (M3) for controlled deformation, as shown in Figure 4.32 and Figure 4.33. The remaining types of plastic hinge can be found in the SAP2000 Manual (2009).
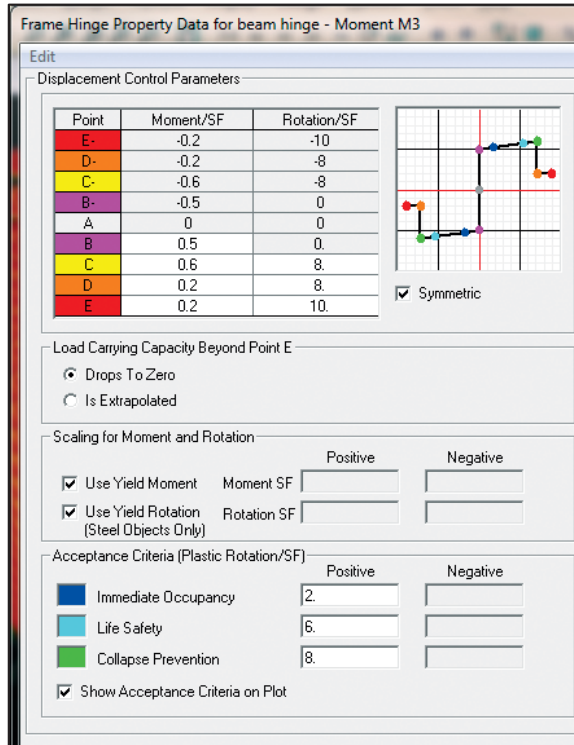
**Fig. 4.33   Steel beam plastic hinge moment–rotation curve.
SAP2000 screenshot reprinted with permission of CSI.**

There also some default plastic hinges that can be selected by the user, which will be introduced in Step 3.

Step 3: Assigning hinge properties

We will assign Default PMM hinges to each end of the moment frame columns and the braced frame columns. We will assign Default M3 hinges to each end of the moment frame beams. We will assign Default P to the centre of each brace.

- From the menu choose *Select*, and then *Groups* from the submenu. This will display the Select Groups dialog box as in Figure 4.34.
- In Figure 4.35, select the groups you want to assign the plastic hinge to. Then from the *Assign* menu choose *Frame*, and then *Hinges*.
- Then define the *Relative Distance* as shown Figure 4.35. Using similar methods, the hinges can be assigned to the required members.
- You can also assign the *Auto Hinge* property as shown Figure 4.36.
- Click on *Modify/Show Auto Hinge Assignment Data*; you can define it from the FEMA356 Table as shown in Figure 4.37.
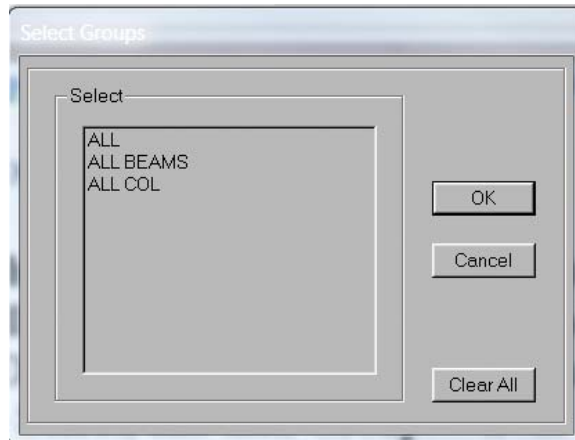
Fig. 4.34   Select the *Groups* dialog box.
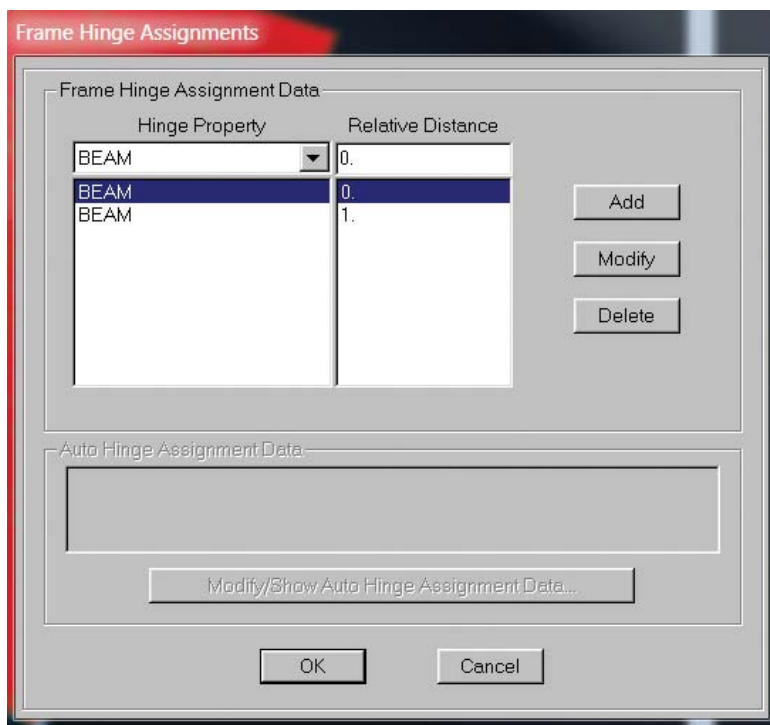**SAP2000 screenshot reprinted with permission of CSI.**

Fig. 4.35   Defining the beam hinge.
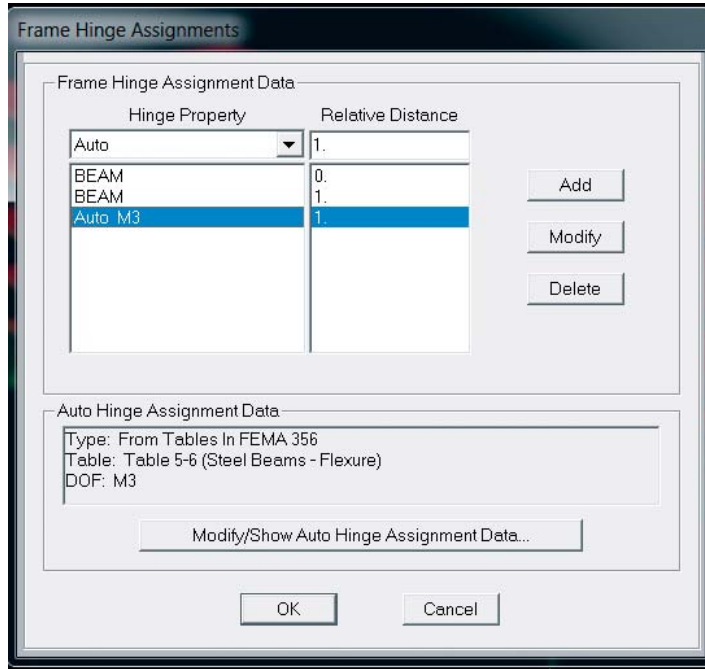**SAP2000 screenshot reprinted with permission of CSI.**

Fig. 4.36 Defining the auto hinge.
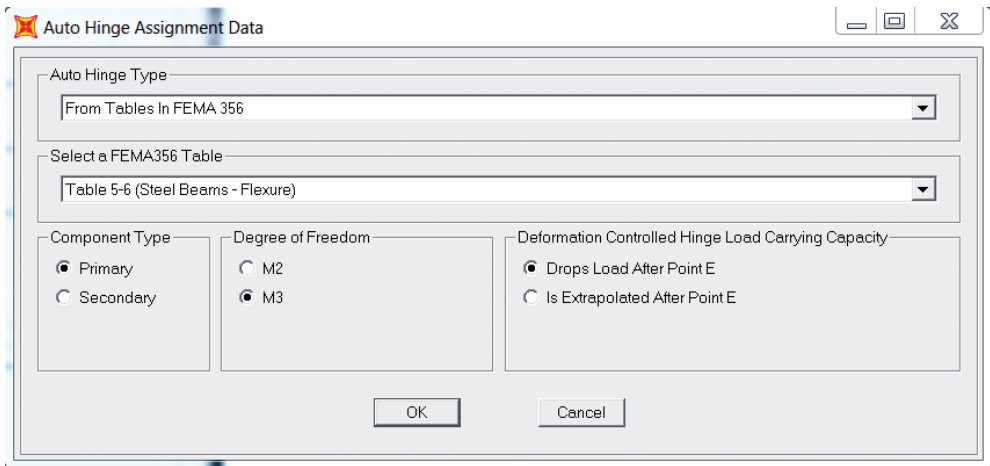SAP2000 screenshot reprinted with permission of CSI.

Fig. 4.37 Defining the auto hinge (2).
SAP2000 screenshot reprinted with permission of CSI.

Step 4: Defining static push-over cases

- In the *Define Load Case* tab, click *Add New Load Case*, as shown in Figure 4.38.
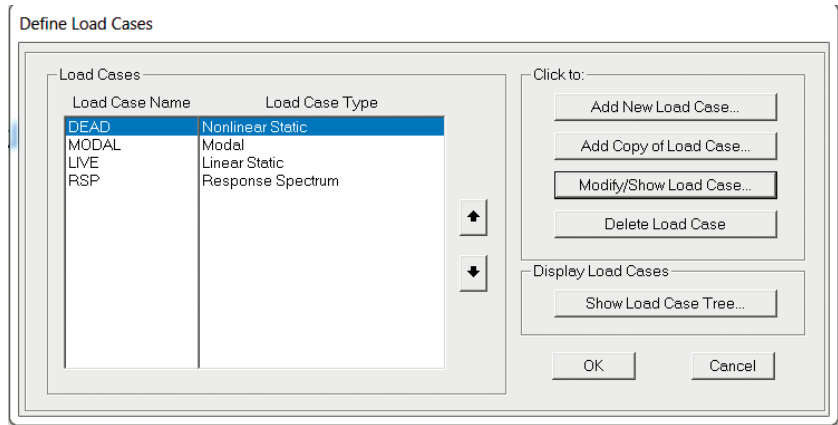- Choose the parameters as shown Figure 4.39.

Fig. 4.38   Load case.
SAP2000 screenshot reprinted with permission of CSI.
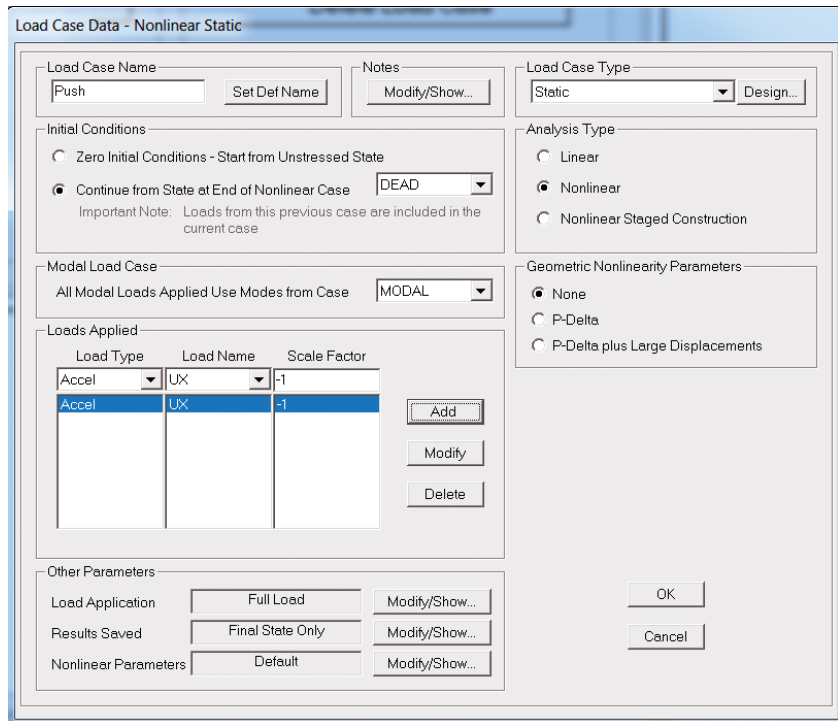

Fig. 4.39   Defining the non-linear static push over case.
SAP2000 screenshot reprinted with permission of CSI.

- Click *Modify/Show Load Application* in the window shown in Figure 4.39, choose the parameters as shown in Figure 4.40, this is to apply an incremental displacement at joint 6 using the displacement control method, which is normally the top of the building.

Fig. 4.40   Choosing the displacement control method.
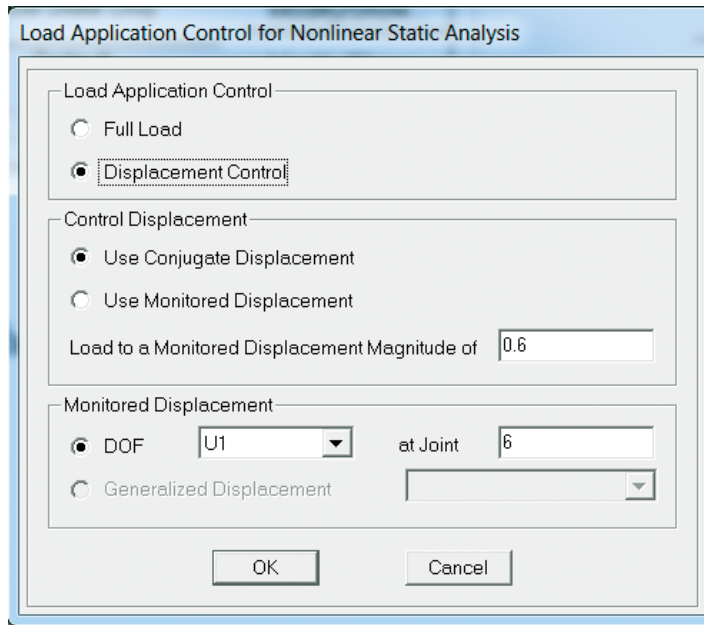**SAP2000 screenshot reprinted with permission of CSI.**
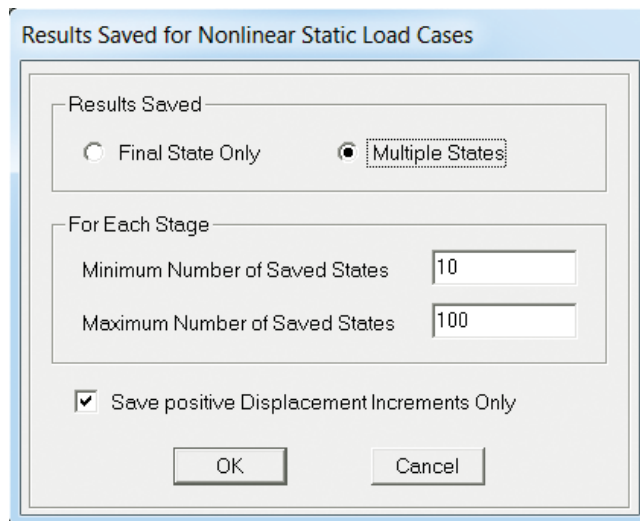
Fig. 4.41   Defining *Result Saved* options.
SAP2000 screenshot reprinted with permission of CSI.

Fig. 4.42   Choosing the push over step.
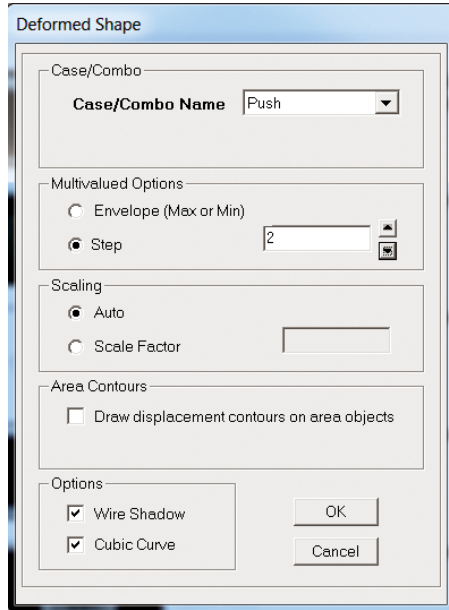**SAP2000 screenshot reprinted with permission of CSI.**
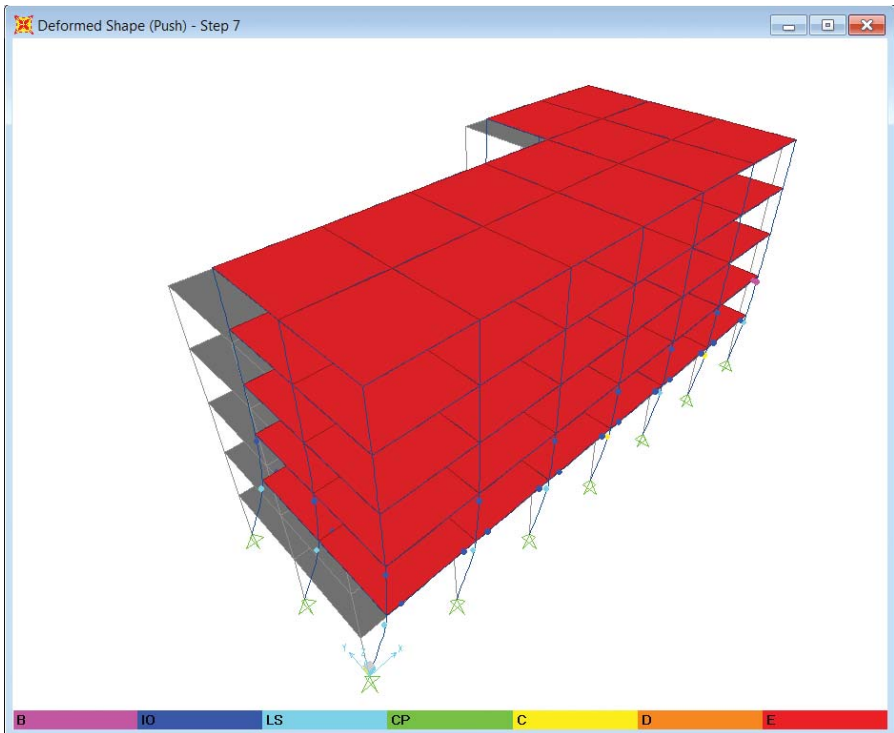

Fig. 4.43   Plastic hinge development.
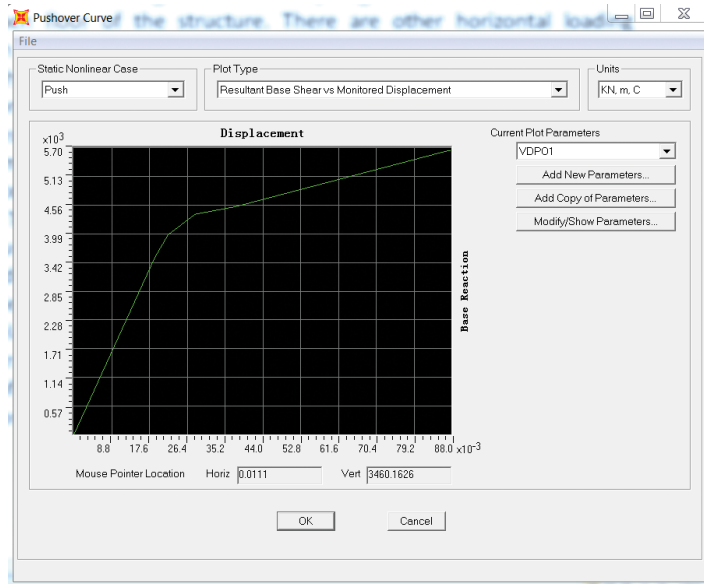**SAP2000 screenshot reprinted with permission of CSI.**

Fig. 4.44   Base shear curve.
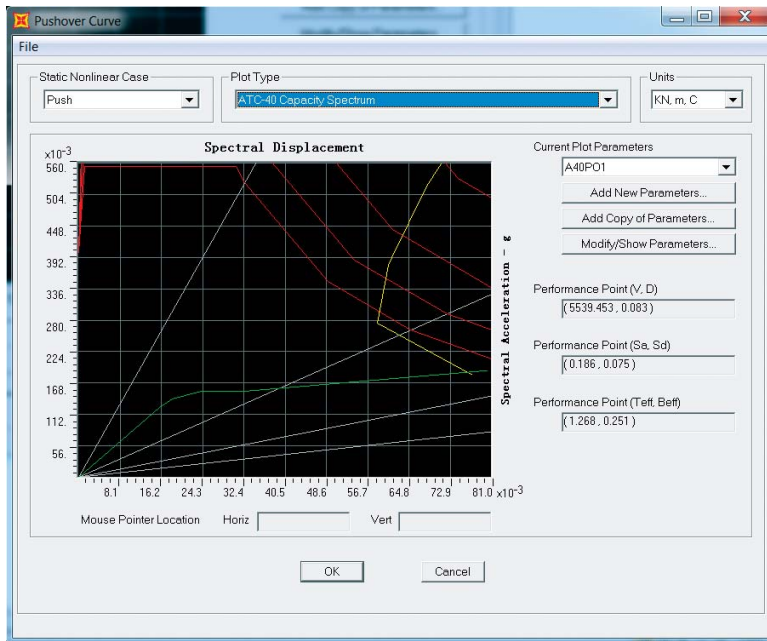**SAP2000 screenshot reprinted with permission of CSI.**

Fig. 4.45   Capacity curve.
SAP2000 screenshot reprinted with permission of CSI.

- Also click *Modify/Show Result Saved* in the window shown in Figure 4.39, and set the parameters as shown in Figure 4.41.

Step 5: Result interpretation

- After analysis, click on *Deformed Shape* in the menu, choose *Push Case*, as shown in Figure 4.42.
- You can see the plastic hinge development in the building in Figure 4.43.
- Click on *Display*, *Show Static Push Over Curve*. You can see the base shear curve in Figure 4.44, or the capacity curve in Figure 4.45.

References

Chopra A K. 2006. *Dynamics of Structure*, 3rd edition, Prentice Hall International Series in Civil Engineering and Engineering Mechanics

Clough R.W. and Penzien J. 1993 *Dynamics of Structures*, 2nd edition, McGraw Hill.

Tedesco J.W., Mcdougal W.G., Ross, C.A. 1999. *Structural Dynamic Theory and Applications*, Addison Wesley Longman, Inc.

Codes and building regulations

ASCE/SEI 7-10 2010. Minimum Design Loads for Buildings and Other Structures, American Society of Civil Engineering.

EN 1998-1: 2004. Design of Structures for Earthquake Resistance. Part 1: General Rules, Seismic Actions and Rules for Buildings. (Eurocode 8). CEN, European Committee for Standardisation.

Software and manuals

SAP2000 Theory Manual. 2009. Version 14, Copyright Computers and Structures, Inc. Berkeley, California, USA.

5 Progressive collapse analysis

5.1 Introduction

Nowadays, increasing numbers of tall building design projects require more attention to the design of the building against progressive collapse. In the UK, special requirement has been made for tall building designs. For a building over 300 m, a separate progressive collapse analysis report is required to be submitted and checked by the building control department. Therefore, in this chapter, the detailed progressive analysis and design method will be introduced. The modelling method of progressive collapse analysis using SAP2000 will also be covered.

5.2 Design guidance for progressive collapse analysis

Progressive collapse first attracted the attention of engineers from the structural failure of a 22-storey apartment building at Ronan Point, London, UK, in 1968. Progressive collapse refers to the action resulting from the failure of one structural element that leads to the failure of further similar elements, which may result in entire structural collapse in some cases (ASCE, 2005). Vehicular collision, aircraft impact and gas explosions are some examples of the potential hazards and abnormal loads that can produce such an event (NIST, 2007).

The United Kingdom was the first country in the world to publish design guidance for preventing the progressive collapse of the building. The UK Building Regulations (2004) has led with requirements for the avoidance of disproportionate collapse. These requirements are refined in material-specific design codes BS5950 (2001) for structural steelwork. It can be described as: (i) prescriptive 'tying force' provisions that are deemed sufficient for the avoidance of disproportionate collapse; (ii) 'notional member removal' provisions that need only be considered if the tying force requirements cannot be satisfied; and (iii) 'key element' provisions applied to members whose notional removal causes damage exceeding prescribed limits. According to this design guidance, during the design process, the engineers should make sure that all the structures to be designed comply with BS 5950-1:2000 clause 2.4.5 (2001) Structural Integrity. Adequate ties will be incorporated into the frame to reduce the possibility of progressive collapse as required by the building regulation. The 'Key Element' will be designed for sustaining an accidental design loading of 34 kN/m^2.

The United States was also one of the first countries in the world to publish detailed design guidance on preventing progressive collapse in building design. The Department of Defence (DoD) (2010) and the General Services Administration (GSA) (2003) provide detailed information and guidelines regarding methodologies to resist progressive collapse of building structures. Both employ the alternate path method (APM) to ensure that structural systems have adequate resistance to progressive collapse. APM is a threat-independent methodology, which means that it does not consider the type of triggering event (such as fire, blast etc.) but rather the response of the structure after the triggering event has destroyed critical structural members. If one component fails, alternate paths should be designed to be available for the load such that a general collapse does not occur. Similar to the 'notional member removal', the methodology is generally applied in the context of a 'missing column' scenario to assess the potential for progressive collapse, and is used to check if a building can successfully absorb loss of a critical member. The technique can be used for the design of new buildings or for checking the capacity of an existing structure.

5.3   Risk assessment

There are various triggers to progressive collapse, such as vehicle bombs, fire or gas explosion. Due to the limited database of progressive collapse events, it is not reasonable to assess the probability of occurrence of hazards. Therefore, a more reasonable route is to carry out a risk assessment.

According to DOD (2009), the risk assessment can be carried out based on the category of the buildings. It reduces to a consideration of consequences. In general, consequences are measured in terms of human casualties and, therefore, the occupancy of a building or structure is often the most critical issue.

Table 2-1 of DOD (2009) gives detailed information of the occupancy categories. When designing a building to prevent progressive collapse, the engineers can use it to determine the category of the building. Table-2-1 is reproduced as Table 5.1.

5.4   Design and analysis method

The main objective of a progressive collapse analysis is to determine the vulnerability of a structure to progressive collapse and design corresponding mitigation measures to prevent such an occurrence. The design guidance in the UK and USA provides three basic design approaches: an indirect method and two direct methods.

5.4.1   Indirect design method

The indirect method consists of prescriptive design guidance to improve the robustness of the building by stipulating the minimum tying forces for connections, identifying key elements and designing for nominal increased design loading. It designs through the provision of minimum levels of strength, continuity and ductility, and therefore is the basic requirement for preventing progressive collapse of the buildings. This method has been adopted by both BS5990 (2001) and ASCE (2005).

Table 5.1   Occupancy category.

Nature of Occupancy	Occupancy Category
• Buildings in Occupancy CategoryIin \1\ Table 2-2 of UFC 3-301-01./1/ • Low Occupancy Buildings[a]	I
• Buildings in Occupancy Category IIin \1\ Table 2-2 of UFC 3-301-01./1/ • Inhabited buildings with less than 50 personnel, primary gathering buildings, billeting, and high occupancy family housing[a,b]	II
• Buildings in Occupancy Category III in \1\Table 2-2 of UFC 3-301-01./1/	III
• Buildings in Occupancy Category IV in \1\Table 2-2 of UFC 3-301-01./1/	IV
• Buildings in Occupancy Category V in \1\Table 2-2 of UFC 3-301-01./1/	

[a]As defined by UFC 4-010-01 Minimum Antiterrorism Standards for Buildings.
[b]Occupancy Category II is the minimum occupancy category for these buildings, as their population of function may require designation as Occupancy Category III, IV, or V.
United Facilities Criteria-DoD, 4-023-03, Whole Building Design Guide, a program of the National Institute of Building Sciences, 2009, Table 2-1.

5.4.2   *Direct design method*

Direct design methods consist of design guidance to increase the strength of the main structural elements to avoid failure under accepted abnormal loads or making the structure so that it can cover the local failure area. There are two direct design methods.

The first direct method is to assess the capacity of the individual members and also the global structure to withstand the loading applied to the structure under each design level hazard. Thus, check their ability to withstand the hazard within agreed acceptance criteria.

Another direct design method is the well-known alternate load path method. This method consists of a systematic assessment of the hazards that can be applied to each structural element and a judgement of whether it can withstand that hazard level. If it is deemed that the structural element would not survive the hazard level then a structural model of the elements and their surrounding structure is developed. This structure is then assessed with the sudden removal of critical structural members, such as columns, to study the structure's ability to redistribute the forces on the structure without triggering progressive collapse. This method has been widely used in current analysis, most researchers such as Fu (2009) and Kim and Kim (2009) use this method in their research. The best way to implement this method is either through Abaqus® or SAP2000; the detailed modelling method will be introduced in a later section. General Services Administration (GSA) (2003) provides the detailed Guidelines on the Alternate Load Path Method that enables structural engineers to analyse the ability of a structure to withstand progressive collapse.

Table 5.2 Occupancy category and design requirements.

Occupancy Category	Design Requirement
I	No specific requirements
II	Option 1: Tie Forces for the entire structure and Enhanced Local Resistance for the corner and penultimate columns or walls at the first story. OR Option 2: Alternate Path for specified column and wall removal locations.
III	Alternate Path for specified column and wall removal locations; Enhanced Local Resistance for all perimeter first story columns or walls.
IV	Tie Forces; Alternate Path for specified column and wall removal locations; Enhanced Local Resistance for all perimeter first and second story columns or walls.

United Facilities Criteria-DoD, 4-023-03, Whole Building Design Guide, a program of the National Institute of Building Sciences, 2009, Table 2-1.

5.4.3 Selection of design method

The selection of above design methods is mainly based on the occupancy category of the building by design guidance such as British Building Regulations (2000) and DOD (2009). Table 2-2 in DoD (2009) (as shown in Table 5.2) provides detailed requirements of the progressive collapse analysis method base on the occupancy category that the engineers can refer to. Similar requirements are also provided in British Building Regulations (2000).

5.4.4 Structural analysis procedures and acceptance criteria

Based on the above design approach, four main analysis procedures were included in GSA (2003). They are: linear static, linear dynamic, nonlinear static and nonlinear dynamic. In progressive collapse analysis, an engineer should choose the approach based on the size and importance of the facility considered and the degree of structural regularity. The correspondent loading and the acceptance criteria to be used are dependent on the type of analysis adopted.

Load combinations

It is recognised that under certain column removal scenarios the loading present will be less than the ultimate design loading for the structure; therefore, GSA (2003) recommended that the load present on the column removal should be the full dead load plus 0.25 times the live load. However, the structural performance under a possible column removal is influenced by the dynamic characteristics of the structure itself, such as the dynamic amplification effect, which are automatically captured for dynamic procedures but not for static analyses. Therefore it is recommended that for static procedures, the load applied to the structure is increased to two times the value recommended above to be on the safe side. The required load combinations are summarised in Table 5.3.

Table 5.3   **Load combination and acceptance criteria.**

	Load combination	Acceptance criteria
Static Linear	2x(Dead+0.25Live)	Demand capacity Ratio
Static Nonlinear	2x(Dead+0.25Live)	Rotation/Ductility
Dynamic Linear	Dead+0.25Live	Demand capacity Ratio
Dynamic Nonlinear	Dead+0.25Live	Rotation/Ductility

Acceptance criteria and analysis steps for linear procedures

The acceptance criteria for the four analyses are also shown in Table 5.3. For linear elastic procedures, the acceptance criteria are based on the demand capacity ratio (DCR) for each structural member and connection in the design. The use of linear elastic procedures is limited to structures that meet the requirements for irregularities and DCRs. Please refer to DOD (2009) for further details. The DCR is simply the force in each member/connection under the considered scenario divided by the expected ultimate, un-factored capacity of the member/connection considered. It is calculated as follows:

$$DCR = Q_{\text{UDLim}}/Q_{\text{CE}}$$

Where:
Q_{UDLim} are resulting actions (internal forces and moments)
Q_{CE} is expected strength of the component or element, as specified in Chapters 4 to 8 in DOD (2009)

Figure 4.3 of the GS A (2003) contains the DCR limit calculation table for beams and columns, the reader can refer to it to choose the correct DCR ratio in the design.

Here, we are going to introduce the steps of linear elastic static progressive collapse analysis, based on the requirement of DOD (2009) and GSA (2003).

Analysis procedure for linear elastic static progressive collapse analysis

- **Step 1.** Remove a vertical support from the location being considered and conduct a linear static analysis of the structure of the model with the load combination of 2(DL +0.25LL) according to Table 5.3.
- **Step 2.** After initial analysis, determine which members and connections have DCR values that exceed the acceptance criteria provided GSA (2003).

    If the DCR for any member end or connection is exceeded based upon shear force, the member is to be considered a failed member.

    If the flexural DCR values for both ends of a member or its connections, as well as the span itself, are exceeded, the member is to be considered a failed member.
- **Step 3.** Remove failed members from the model, and all dead and live loads associated with failed members should be redistributed to other members in adjacent bays.
- **Step 4.** For a member or connection whose $Q_{\text{UD}}/Q_{\text{CE}}$ ratio exceeds the applicable flexural DCR values, place a hinge at the member end or connection to release the moment. (This can be achieve using analysis program such as SAP2000.)

- **Step 5.** Re-run the analysis and repeat steps until no DCR values are exceeded.
- **Step 6.** Judgement of the collapse of buildings is performed as follows.

  If moments have been re-distributed throughout the entire building (this can be achieved through checking the distribution of plastic hinges in analysis packages such as SAP2000) and DCR values are still exceeded in areas outside of the allowable collapse region, the structure will be considered to have a high potential for progressive collapse.

From the above procedure, we can see that the linear elastic static analysis procedure is quite simple and straightforward. But an engineer needs to repeat the steps several times before reaching a conclusion. Readers can fulfil this analysis through software such as SAP2000 or ETABS.

Acceptance criteria and analysis steps for nonlinear procedures

As shown in Table 5.3, for nonlinear procedures, the acceptance criteria are based on less onerous rotation and ductility demands for the members/connections considered. The analysis can be achieved using such approaches.

Analysis procedure for nonlinear dynamic procedure procedures

- Development of a model (ideally this model could include materially nonlinear, which can be achieved using a general purpose program such as ANSYS or Abaqus® as they can direct simulate the nonlinear feature such as the plasticity without inserting plastic hinge, with a load combination of full dead load plus 0.25 times the live load according to Table 5.3).
- A static step to determine the forces under this load combination of full dead load plus 0.25 times the live load.
- The removal of a column in a dynamic time step which is less than 1/10th the period of the fundamental mode including geometric nonlinearity. (In the author's experience the removal duration is important to achieve reliable analysis result). The determination of the subsequent free dynamic response under the column removal scenario including geometric nonlinearity (if the analysis program allows; programs such as Abaqus® and ANSYS allow geometric nonlinearity) or inserting the plastic hinges (in a program such as SAP2000 etc.).
- The maximum forces, displacements and rotations for each of the members/connections involved in the scenario will be determined from the dynamic analysis for comparison with the acceptance criteria outlined in Table 2.1 of the GSA guidelines (2003).

Summary

It can be seen that the recommendations in Table 5.3 for each type of analysis result in a different degree of inherent conservatism. The actual degree of conservatism will be dependent on the structural form considered. The linear static procedure is easy to implement, the nonlinear dynamic procedure requires more modelling techniques, and therefore, in the next section we are going to demonstrate how to implement it in SAP2000.

5.5 Modelling example of progressive collapse analysis using SAP2000 – nonlinear dynamic procedure

In this section, a modelling example of progressive collapse analysis using SAP2000 will be demonstrated. The nonlinear dynamic procedure will be used here. Therefore, the load combination used here is DL+0.25LL, as required by GSA (2003).

The basic procedure for performing a progressive analysis in SAP2000 is through a time history analysis. The general procedures are as follows:

1. Set up a detailed 3D model.
2. Run an initial static analysis with the load combination DL+0.25LL.
3. Delete the column that will be removed, remember to check the column load before deleting.
4. Replace the column with an upward joint load of the same magnitude as an axial force in the column that was deleted.
5. Run column removal analysis by using a time history function that will remove the load applied in Step 4 over a user defined time period to simulate the removal of the column.

A 5-storey steel composite moment frame building was set up in SAP2000 as shown in Figure 5.1.

A step by step set of instructions is included below:

Step 1: Define removal time history

- Similar to that introduced in Chapter 4, click *Define*, *Function*, *Time History*, and add *User Defined Function* as shown in Figure 5.2.
- Click *Add New Function*.
- A new window will pop up, as shown in Figure 5.3, define the time history function as shown.

Step 2: Define load cases

- Define load pattern.

Define the load pattern as shown in Figure 5.4, add a reaction load pattern called *Reaction*.

- Define load cases.

There are three extra load cases that need to be defined here, they are *Reaction Cases*, *Initial Static Cases*, and *Removal Cases* respectively, as shown in Figure 5.5.

- For the initial static load case, choose the parameters shown in Figure 5.6, the scale factor is chosen according to the required load combination by GSA (2003).
- For the column removal load case, choose the parameters as shown Figure 5.7; remember to set the analysis to continue from the initial static analysis. For the *Number of Output Steps* choose 100, for *Output Time Step Size* 0.01.
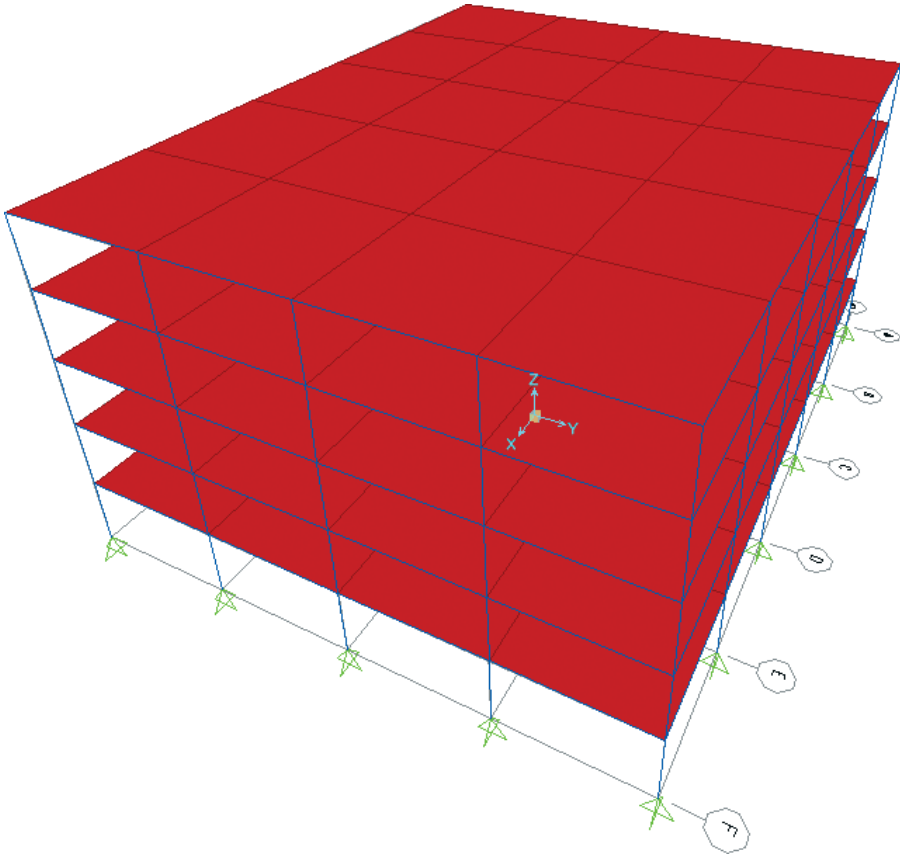
Fig. 5.1   A 5-storey steel composite moment frame building.
SAP2000 screenshot reprinted with permission of CSI.
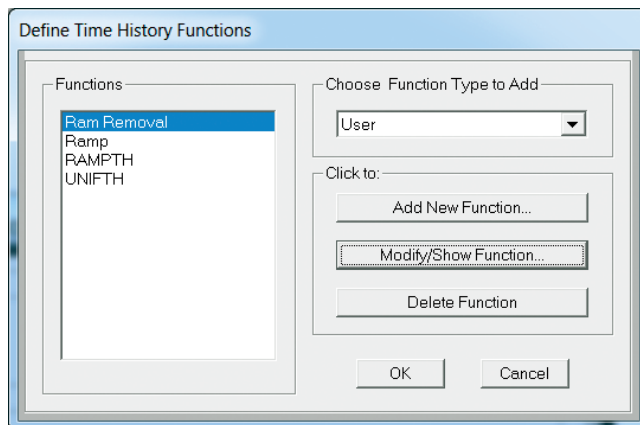
Fig. 5.2   Defining the time history function.
SAP2000 screenshot reprinted with permission of CSI.
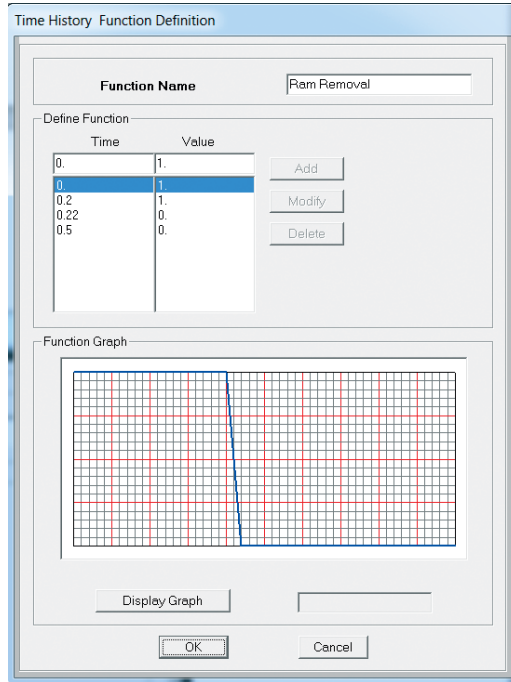
Fig. 5.3   Time history function for removal.
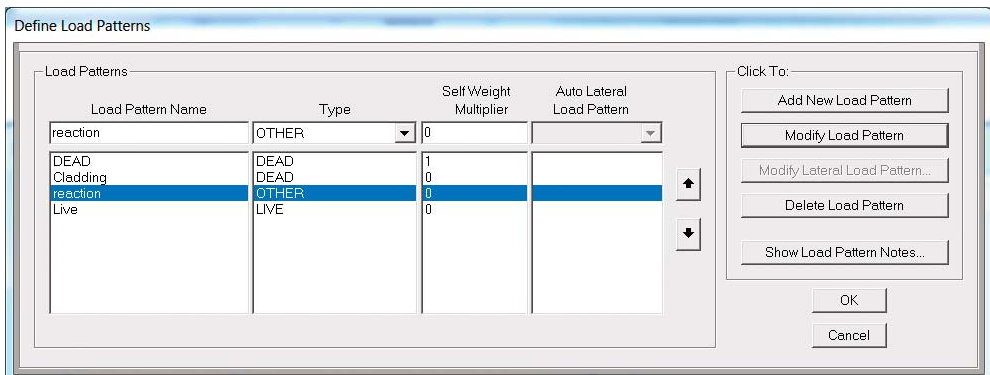**SAP2000 screenshot reprinted with permission of CSI.**

Fig. 5.4   Load pattern.
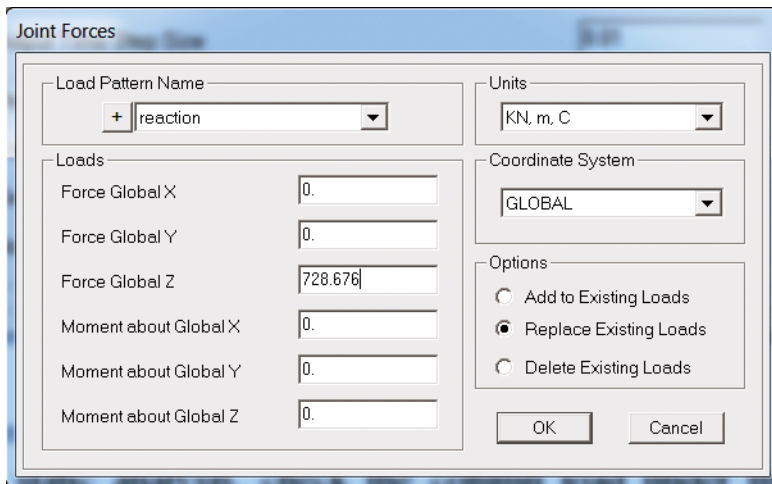**SAP2000 screenshot reprinted with permission of CSI.**

Step 3: Run initial static analysis and apply reversed column load

- Run the static analysis.
- Check the column load under the initial static load case for the targeted removal columns, and then delete the column manually.
- Apply a reversed equivalent value reaction force, make sure to select the load case *Reaction* in Figure 5.8.
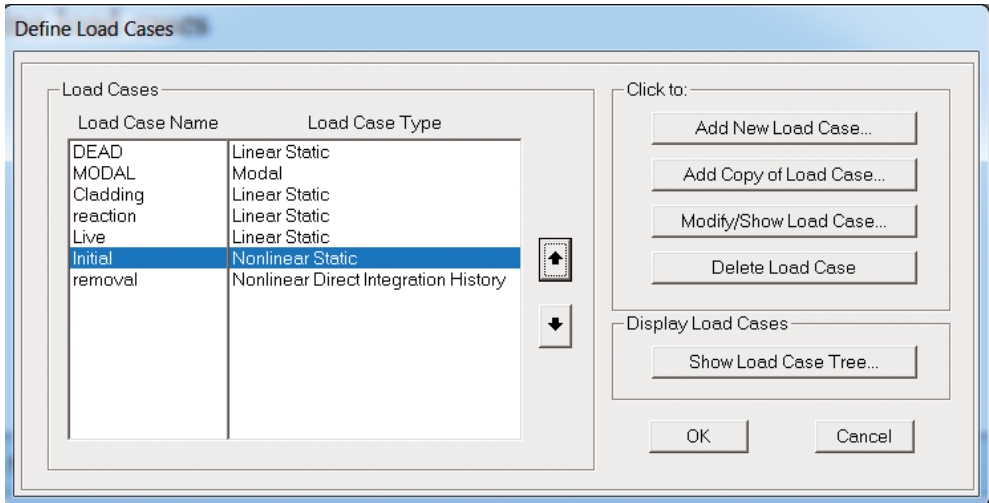
Fig. 5.5   Load cases.
SAP2000 screenshot reprinted with permission of CSI.
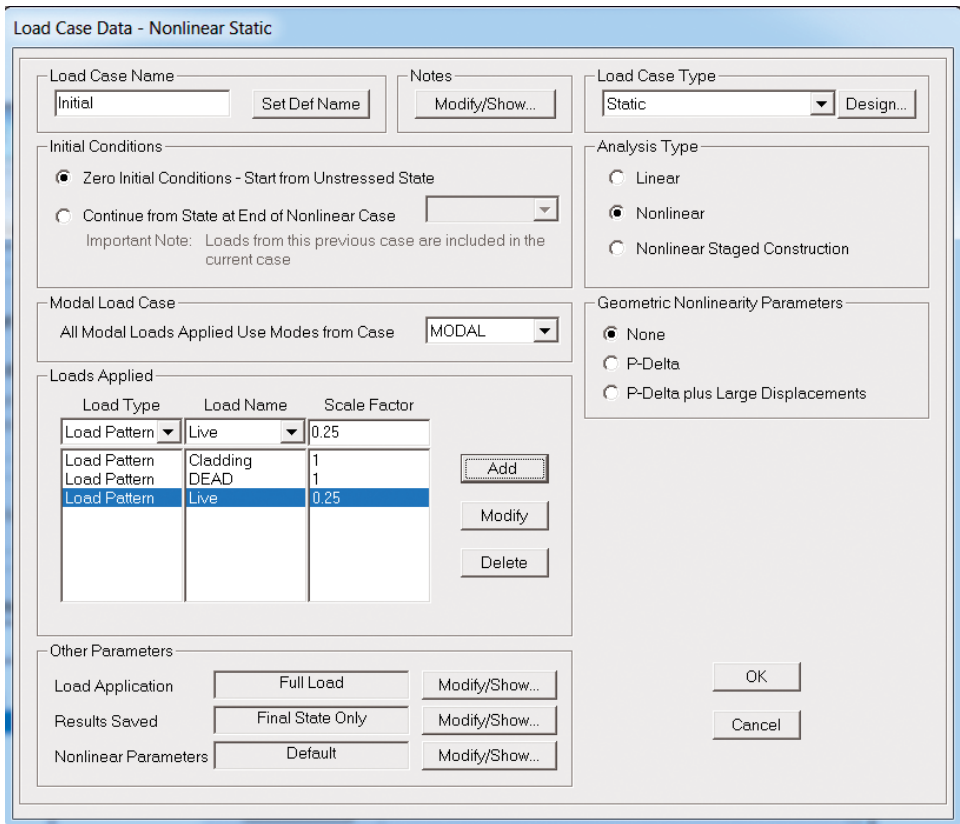
Fig. 5.6   Initial static analysis case.
SAP2000 screenshot reprinted with permission of CSI.
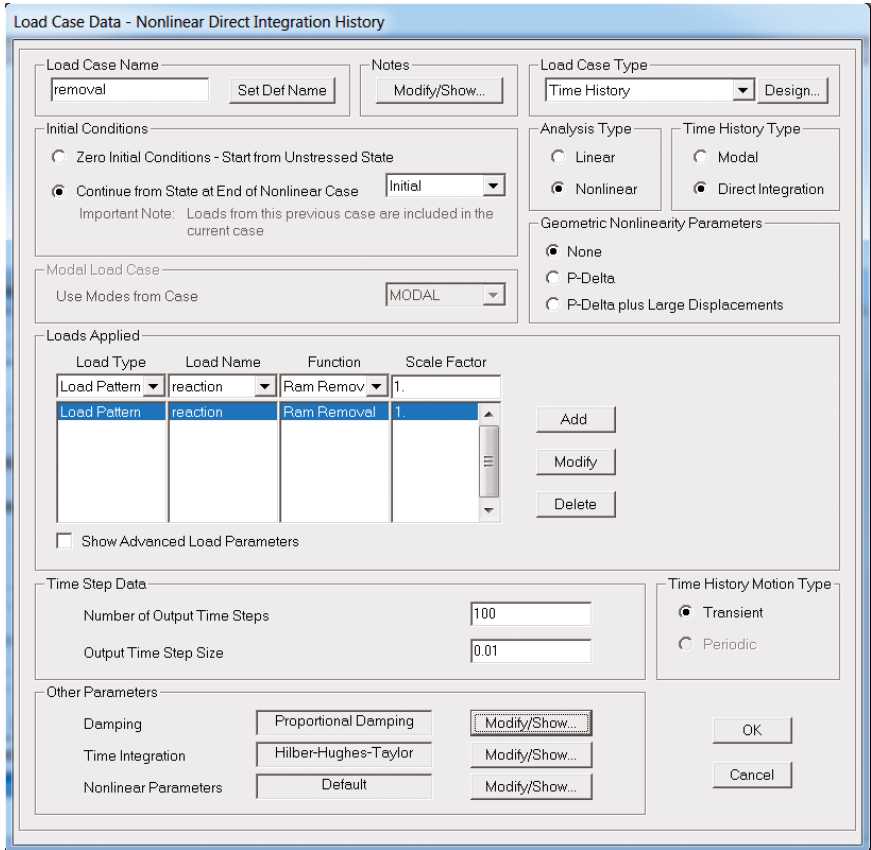
Fig. 5.7   Column removal cases.
SAP2000 screenshot reprinted with permission of CSI.

Fig. 5.8   Defining the reversed column load.
SAP2000 screenshot reprinted with permission of CSI.
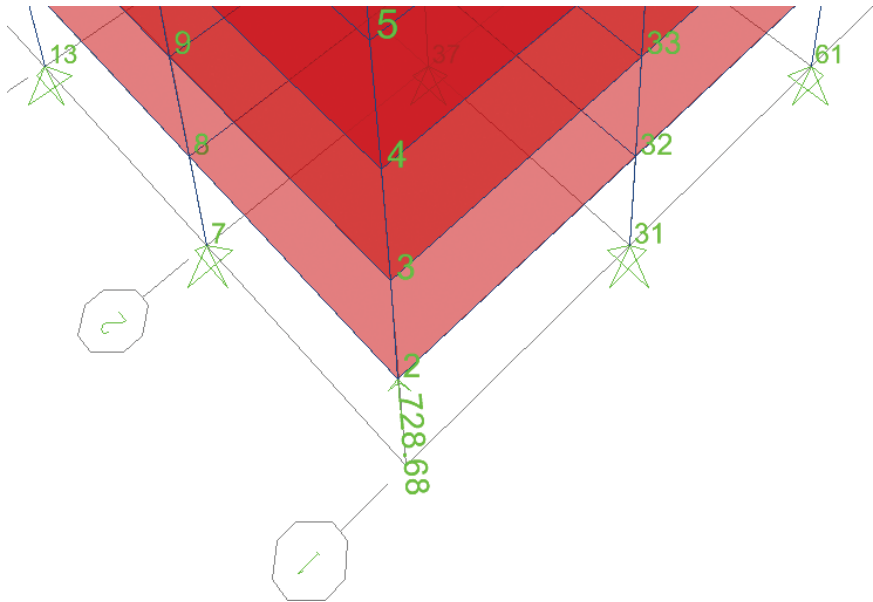
Fig. 5.9 Reversed column load applied to the building.
SAP2000 screenshot reprinted with permission of CSI.

Now the reaction force is applied to the point where the column has been removed, as shown in Figure 5.9.

Step 4: Define the plastic hinges to column and beams

Following the steps in Chapter 4, the plastic hinge can be defined and assigned. For the rules to define the plastic hinges, please refer to GSA (2003) and DOD (2009) for detailed instructions.

Step 5: Run removal analysis

After the plastic hinge definition, run the removal analysis.

Step 6: Result interpretation

- Go to *Display – Show Plot Function*; a new window will pop up as shown in Figure 5.10.
- Click *Define Plot Functions* in Figure 5.10.
- A new window will pop up as shown in Figure 5.11, choose *Add Frame Forces* (you can choose displacement or internal force of the beam or column to investigate: here, we choose the frame force for demonstration).
- Put in the element number (beam or column ID, you can check this in SAP2000) you want to investigate, as shown in Figure 5.12.
- And select the axial force or any other internal force you want to check.
- Display the plot function traces (as introduced in Chapter 4).
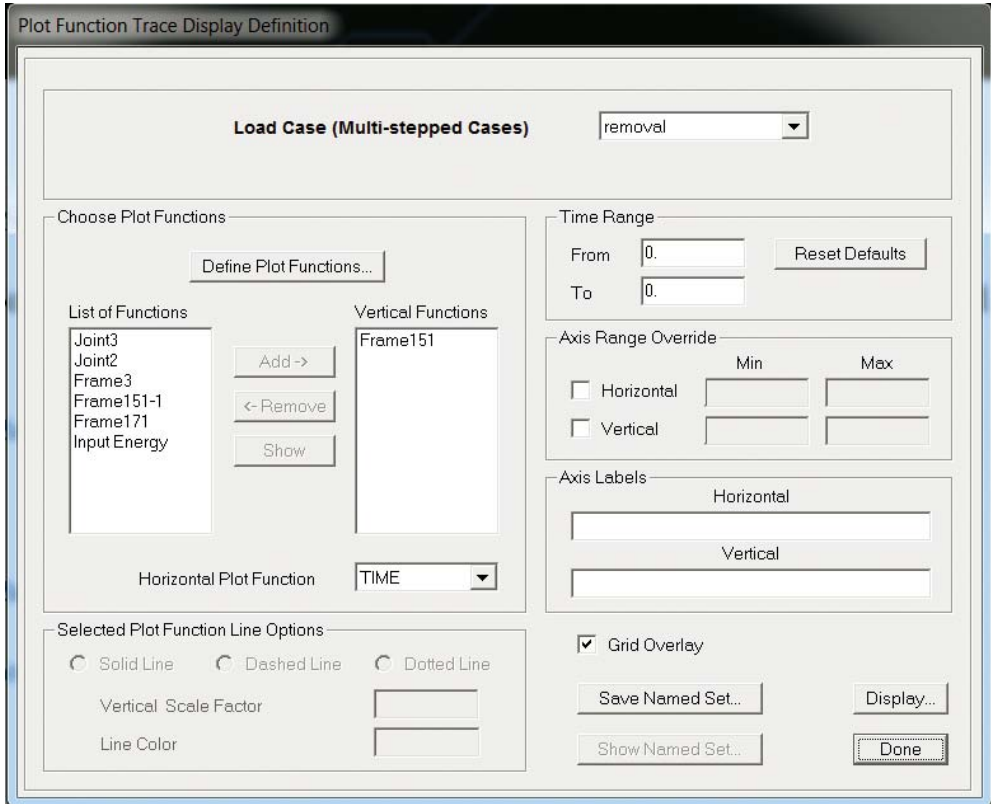
Fig. 5.10   Plot time history.
SAP2000 screenshot reprinted with permission of CSI.
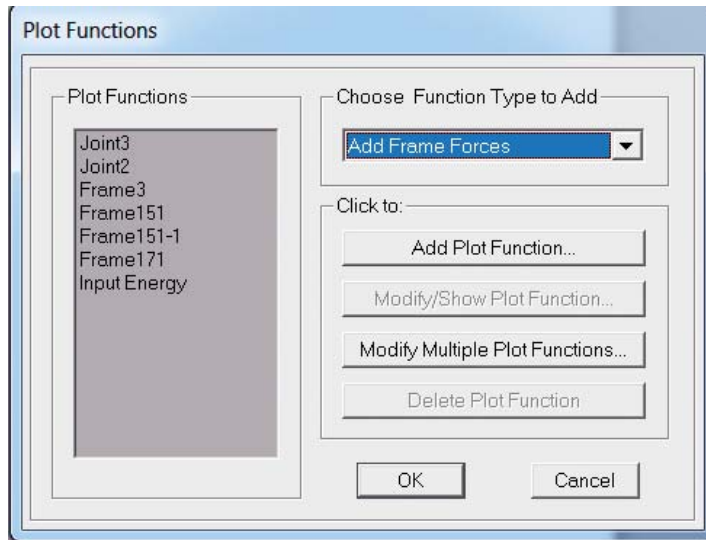
Fig. 5.11   Add frame force.
SAP2000 screenshot reprinted with permission of CSI.
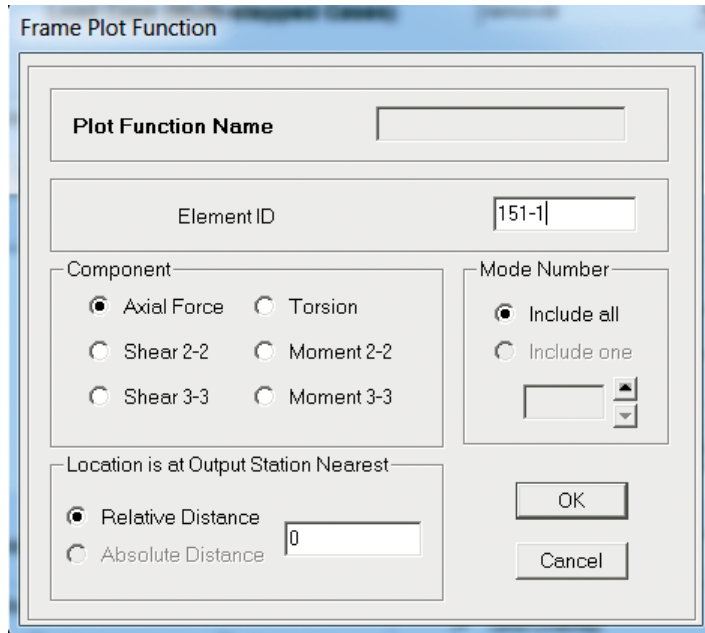
Fig. 5.12   Choosing the component and frame ID.
SAP2000 screenshot reprinted with permission of CSI.
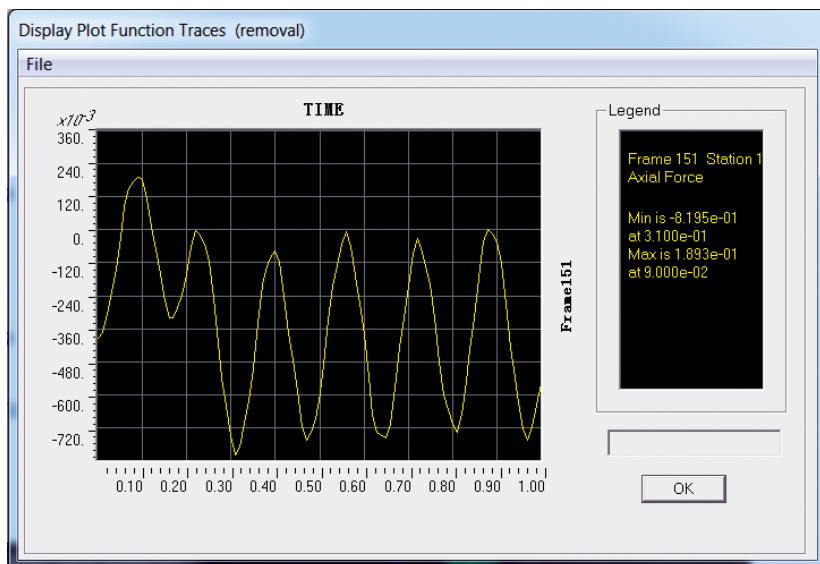
Fig. 5.13   Time history for axial force of Frame 151.
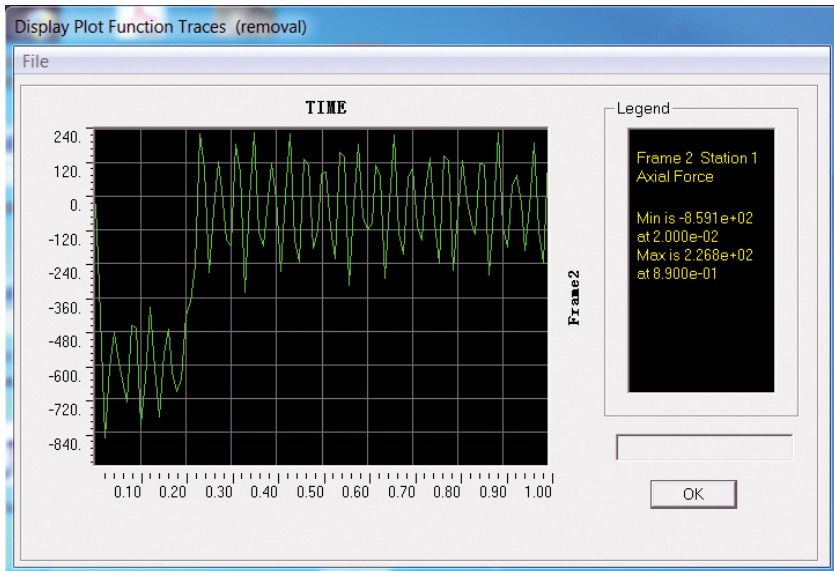
Fig. 5.14   Time history for axial force of Frame 2.
SAP2000 screenshot reprinted with permission of CSI.

Figure 5.13 and Figure 5.14 show the result of the time history for Frame 151 and Frame 2: we can clearly see the dynamic response of the structural member after the column removal.

References

Fu, F. 2009. "Progressive collapse analysis of high-rise building with 3-D finite element modelling method", *Journal of Constructional Steel Research*, 65: 1269–1278.

Kim J and Kim T. 2009. "Assessment of progressive collapse-resisting capacity of steel moment frames", *Journal of Constructional Steel Research*, 65, 1: 169–179.

Codes and building regulations

ASCE. 2005. SEI/ASCE 7-05 Minimum Design Loads for Buildings and Other Structures. Washington, DC: American Society of Civil Engineers

British Standards Institution. 2001. BS 5950: Structural use of steelwork in buildings, Part 1: Code of practice for design — rolled and welded sections, London (UK)

GSA. 2003. Progressive collapse analysis and design guidelines for new federal office buildings and major modernization projects. The U.S. General Services Administration.

NIST. 2007. Best Practices for Reducing the Potential for Progressive Collapse in Buildings National Institute of Standards and Technology, Technology Administration, U.S. Department of Commerce.

Office of the Deputy Prime Minister. 2004. The Building Regulations 2000, Part A, Schedule 1: A3, Disproportionate collapse. London, UK.

Unified Facilities Criteria, Department of Defense. 2009. UFC 4-023-03, Design of Buildings to Resist Progressive Collapse 14 July 2009 with Change 2.

6 Blast and impact loading

6.1 Introduction

An explosion can cause damage to the building's structural frame, which may cause structural collapse. Terrorist attacks such as 911 and the car bomb attack in Oklahoma City in 1995 (Astaneh-Asl, 2003) focused great attention on the buildings' designers. In current design practice, designing buildings against blast loading has increasingly become a demand placed on structural engineers. The special design requirements for buildings to resist blast load have become more and more important for the safety of the occupants. For all these reasons, it is therefore imperative for a structural engineer to obtain basic knowledge on design of buildings under blast load, and the simplified modelling method for blast analysis.

6.2 Fundamentals of blast loading

Over the past decades methods of structural analysis and design guidance to resist blast loads have developed. The UK SCI publication 244 (Yandzio and Gough, 1999) provides guidance on the design of commercial and public buildings where there is a requirement to provide protection against the effects of explosions caused by the detonation of explosives. A philosophy for the design of buildings to reduce the effects of an attack is introduced and a design procedure is proposed. The robustness of buildings and the prevention of disproportionate collapse are also discussed. In the United States, The Technical Manual, Army TM5-1300 (1990) is one of the most detailed design guidances and introduces the design of buildings against blast loading. FEMA 426 (2003) provides design measurements to reduce physical damage to the structural and non-structural components of the building and related infrastructures during conventional bomb attacks, as well as attacks using chemical, biological and CBR agents.

6.2.1 Basic design principles

According to SCI publication 244 (1999) and TM5-1300 (1990), the basic design principles of building against blast load is to minimise the likelihood and magnitude of threat, which can be achieved by adopting preventive measures that discourage or

impede an attack. One should also design the building to protect the people and assets from the effect of blast waves and projectiles by providing the occupants with either a safe area or an effective escape route and assembly area.

In the event of fire, design measures should also prevent it burning out of control; detailed guidance will be introduced in Chapter 7.

In terms of structural design, the building should also be designed to prevent progressive collapse. This will be achieved by providing sufficient ductility and redundancy, such as an alternative load path in the structural design.

6.2.2   Major blast attack regimes

There are many ways in which an explosive device may deliver an attack. As stated in SCI (1999), there are several devices such as vehicle bombs, package bombs, mortar bombs, culvert bombs and incendiary devices. For detailed information, please refer to Table 6.1 from SCI (1999).

6.2.3   Blast load characteristics

Explosions can be classified into: unconfined explosions, confined explosions and explosives attached to a structure. Unconfined explosions can further be subdivided into: free-air burst, air burst and surface burst.

In current design guidance, such as SCI (1999) and TM5-1300 (1990), blast loadings are evaluated using empirical relationships based on the principle of the scaling law. These relationships assume that there are no obstacles between the charge and the target. In real situations, the actual blast loads can either be reduced due to shadowing by other buildings or can be enhanced due to the presence of other buildings in the vicinity.

6.2.4   Principle of the scaling law

The principle of the scaling law is used extensively to determine blast-wave characteristics in most design guidance such as SCI (1999) and TM5-1300 (1990). It is based on the conservation of momentum and geometric similarity. The empirical relationship, formulated independently by Hopkinson (1915) and Cranz (1926), is described as a cube-root scaling law and is defined as:

$$Z = \frac{R}{W^{\frac{1}{3}}} \tag{6.1}$$

where:
Z is the scaled distance, with units (m/kg-1/3)
R is the range from the centre of the charge
W is the mass of spherical TNT charge (kg).

The progression of a free-air burst is best represented by a pressure time history curve. The magnitude of the pressure caused by a blast wave is usually quoted as an overpressure (the pressure increase relative to ambient pressure). The characteristics of the blast wave used for calculation purposes are shown as a blast overpressure–time curve presented in Figure 6.1.
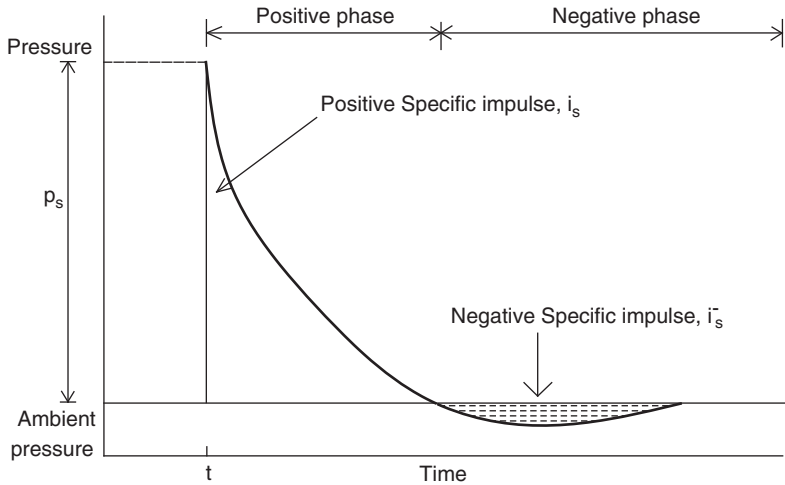
Fig. 6.1 Blast overpressure profile.
AutoCAD.

A blast wave that impinges on a solid surface of a building (or on a dense medium) will be reflected. These reflections, particularly in built-up areas, create complex loading conditions and therefore should be included in the blast overpressure profile as well. The reader can refer to SCI (1999) for further information.

6.2.5   Simplification of the blast load profile

It can be seen from Figure 6.1 that the actual blast pressure profile is not that straightforward to use, and can be reasonably simplified. It is usually adequate to assume that the decay (and growth) of blast overpressure is linear. For the positive overpressure phase, a simplification is made where the impulse of the positive phase of the blast is preserved and the decay of overpressure is assumed to be linear. This simplification is shown in Figure 6.2.
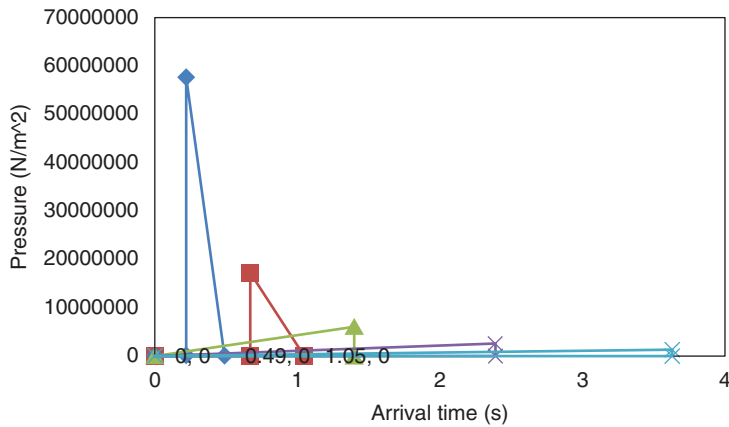
Fig. 6.2 Blast overpressure profile at different arrival times.

6.2.6 Material behaviours at high strain-rate

In the event of blast loading, due to the high rates of strain the dynamic ultimate strength increases and can be much greater than the static ultimate strength. The mechanical properties under blast loading are affected noticeably by the rate at which straining takes place.

The dynamic design yield stress of the steel $F_{y,des}$ for bending is given by SCI (1999):

$$F_{y,des} = a(DIF)F_y \hspace{4cm} (6.2)$$

Where:

a   is a factor that takes into account the fact that the yield stress of a structural component is generally higher than the minimum specified value, for S275 and S355 steels a =1.10.

DIF   are the Dynamic increase factors for structural steels that can be checked in Table 9.1 of SCI (1999).

Similarly, the dynamic strength of the reinforced concrete can be worked out following Equation 6.2. The dynamic increase factors for concrete are calculated following Table 9.3 of SCI (1999).

6.2.7 Dynamic response and pressure impulse diagrams

Blast load imparts energy to the structure, causing it to behave dynamically. How the structure (or structural element) responds to blast loading is linked closely to the ratio between its natural period and the duration of the blast. Three response regimes are defined as:

1. impulsive
2. quasi-static
3. dynamic.

Based on the above three responses, SCI Publication 244 (1999) further defines it based on the natural period of the structure as follows:

Impulsive $t_d/T < 0.4$
Dynamic $0.4 < t_d/T < 2$
Quasi-static $t_d/T > 2$

Where:
t_d is the duration of the blast load
T is the natural period of vibration of the structural element

It is important to note that for any particular blast wave, the response could be impulsive for one structure but quasi-static for another because of the different natural period of vibration of each structure.

In real design practice, engineers can design the building under blast loading using the analytically derived so called pressure–impulse diagrams, which are derived from experimental evidence or real-life events. However, due to their complexity, they require quite a lot of manual calculations. Therefore, computer programs are an effective tool to perform blast analysis. In the next section, we are going to introduce how to model the blast or impact effect in Abaqus®.

6.3   Introduction of SPH theory

In the analysis of the blast or impact effect, the duration of the blast loads applied to the structure and the effect of flying debris are difficult to model using the conventional finite element method (FEM). FEM will have difficulty simulating solid material under extreme situations such as high velocity impact and fragmentation. This is because the FEM is based on the variation principle and continuum mechanics. Finite elements are joined together; no matter how large the extreme load is the elements will not separate. So, the FEM needs special consideration and modification when used to model progressive collapse problems involving strong nonlinearities and discontinuities.

To solve this problem, one of the best options is to use the mesh free Lagrangian method of space discretisation, the smoothed particle hydrodynamics (SPH) method, which is a pure Lagrangian method originally developed for astrophysical computations (Lucy 1977; Monaghan 1992) and later extended to model a wide range of problems. Unlike the traditional Eulerian approach, the SPH does not use a grid to calculate spatial variables and derivatives. The basic concept of SPH is that a particle is fundamental in the Lagrangian description and the motion of a continuum can be represented with very high accuracy by simulating the movement of an increasingly large number of such particles. Through the use of integral interpolants, the dependent field variables are expressed by integrals that are approximated by summation interpolants over neighbouring particles. Similarly, spatial derivatives at each particle location are evaluated by summation interpolants. The derivatives are evaluated based on the derivatives of the kernel function, which are obtained analytically. Compared with other methods, material boundaries and interfaces in the SPH are better defined and material separation is naturally handled. Therefore, SPH provides an excellent tool for simulating the physics of dynamic events in solids and fluids where large deformations of the problem domain are expected. It is particularly well suited to problems in computational solid mechanics that result in complex fracture paths, since in a random distribution of nodes there is no preferential path to crack propagation. Therefore, it provides a more accurate simulation of dynamic fracture and fragmentation of brittle solids (Libersky et al. 1993). Figure 6.3 demonstrates a procedure where an impact loading effect is simulated using the SPH method.

In the SPH method, the state of a system is represented by arbitrarily distributed particles, which carry all the computational information including mass and velocity as well as elastic modulus. Using the concept of kernel and particle approximation, the discretised SPH equations of motion for numerical simulation of blast analysis
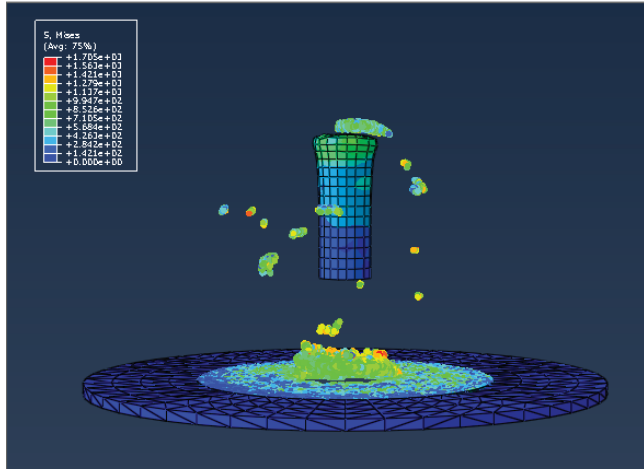
Fig. 6.3   Example of simulation using the SPH method.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

are given as follows (Monaghan 1992):

$$\frac{D\rho}{Dt}(x_i) = \sum_{j=1}^{N} m_j(v(x_j) - v(x_i))A_{ij} \tag{6.3}$$

$$\frac{Dv^\alpha}{Dt}(x_i) = \sum_{j=1}^{N} m_j \left(\frac{\sigma^{\alpha\beta}(x_j)}{\rho_i^2}A_{ij} - \frac{\sigma^{\alpha\beta}(x_i)}{\rho_j^2}A_{ji} \right) \tag{6.4}$$

$$\frac{DE}{Dt}(x_i) = \frac{P_i}{\rho_i^2} \sum_{j=1}^{N} m_j(v(x_j) - v(x_i))A_{ij} \tag{6.5}$$

$$A_{ij} = \frac{x_i - x_j}{r_{ij}} \frac{\partial W_{ij}}{\partial r_{ij}} \tag{6.6}$$

Equation 6.3 is the conservation equation of mass.
Equation 6.4 is the conservation equation of momentum.
Equation 6.5 is the conservation equation of energy.
Where:
ρ is density
v is spatial velocity
E is energy
x is spatial location
t is time
i is the reference particle
j is the neighbouring particle
α and β are spatial dimensions
p is pressure

τ is shear stress
σ is total stress
r_{ij} is the distance from particle i to j
W is the SPH kernel function and can have different forms

The above equations calculate the evolution of each particle's density, acceleration, energy and displacement. By treating the solid structures as composed of a number of individual particles and tracking the motion of all these particles following the above equations, the structure's cracking and fragmentation can be easily analysed. The above equations can be integrated by using numerical methods such as the leapfrog technique.

6.4   Modelling examples of impact loading analysis using the coupled SPH and FEA method in Abaqus®

In this section, we will introduce how to use the coupled SPH and FEA method to model a projectile impact on a plate in Abaqus®. Although the SPH has advantages in simulating extreme deformation, it is not as good as FEM for exploring the structural response and is also relatively more expensive in terms of computational cost. Therefore, a combination of these two numerical methods provides a better option. The latest version of Abaqus® has developed a new function that can link the SPH analysis to the FEM analysis, thereby greatly enhancing accuracy in the analysis of the impact loading in addition to cost effective computation time and an easy to interpolate analysis result.

The method can use any of the materials in Abaqus®/Explicit (including user materials). You can specify initial conditions and boundary conditions as for any other Lagrangian model. Contact interactions with other Lagrangian bodies are also allowed in Abaqus® (2013).

6.4.1   Modelling technique

In Abaqus®, the smoothed particle hydrodynamic method is implemented using PC3D elements. It is a 1-node particle element that utilises existing functionality in Abaqus® to reference element-related features such as materials, initial conditions, distributed loads and visualisation. Abaqus® provides an alternative to define conventional continuum finite element types C3D8R, C3D6 or C3D4 and automatically converts them to particle elements at the beginning or during the analysis.

Contact interactions between SPH and FE mesh

Abaqus® allows contact between the particles and other finite element meshed bodies. The contact interaction is the same as any contact interaction between a node-based surface (associated with the particles) and an element-based or analytical surface. Both general contact and contact pairs can be used. All interaction types and formulations available for contact involving a node-based surface are allowed, including cohesive behaviour. Different contact properties can be assigned via the usual options.

Finite element conversion to SPH particles

In ABAQUS, the element conversion to particle functionality is not active by default. You can control the number of particles generated per parent element and choose between one of four criteria to specify when the conversion is to be triggered; this will be introduced in the modelling example.

For further information on the SPH modelling method, please refer to Abaqus® (2013).

6.4.2 Modelling example

The following example illustrates a smoothed particle hydrodynamic analysis of the impact interaction between a high-velocity cylinder and a solid plate using the conversion technique.

A solid plate, of which the central part is modelled using the SPH technique, is subjected to an impact by a high-velocity cylindrical rigid object. After the impact, the area close to the centre of the plate first undergoes a large deformation and then breaks apart. Eventually, the projectile perforates the plate. The failure of a rate-dependent elastic-plastic material upon impact of a high-speed projectile is modelled with the SPH method, which is hard to represent by other modelling techniques. Contact interaction between the PC3D elements and FE elements is also demonstrated.

A cylinder with the length and radius of 30 mm and 10 mm, respectively, is modelled as shown in Figure 6.4. The initial speed of the projectile is set to 1000 m/s.

A circular solid plate is modelled using C3D8R elements with a diameter of 100 mm as shown in Figure 6.5. A circular part with a radius of 57 mm in the centre of the plate
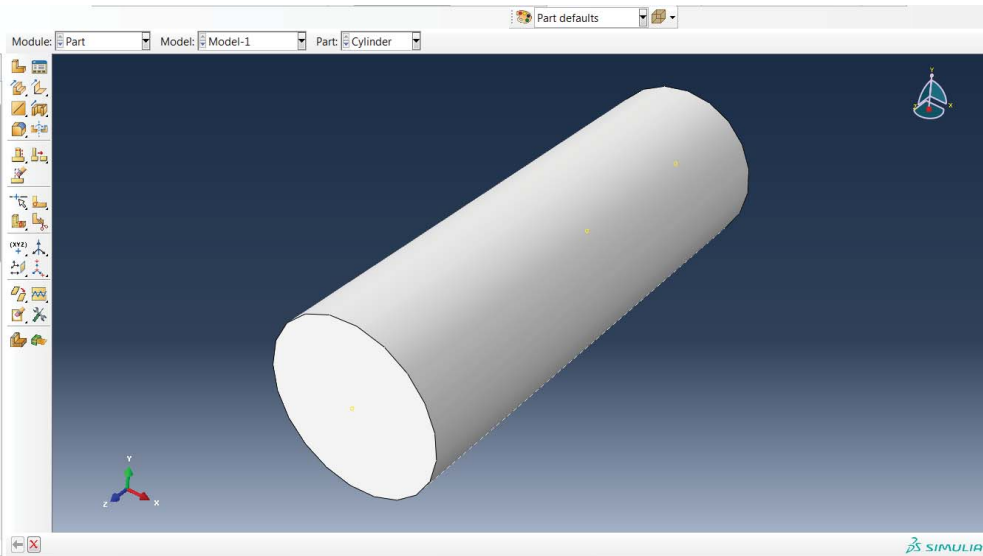
Fig. 6.4   Cylinder part.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
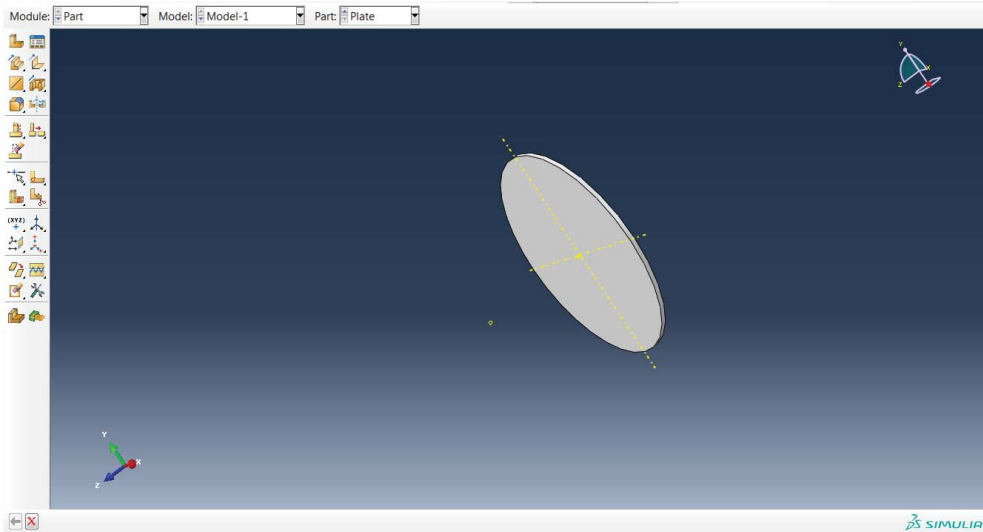
Fig. 6.5   Plate part.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

is modelled using C3D8R elements that will convert to four particles per element at the beginning of the analysis based on a time-based criterion.

The material used for the plate and the cylinder is steel with Young's modulus 2.1×10^5 MPa, Poisson's ratio 0.3, and density 7.8×10^{-9} tonne/mm³. The plate is modelled as an elastic-plastic material with rate-dependent hardening. Ductile and shear damage are evolved based on an energy criterion. The interaction between the rigid projectile and the solid plate is defined using contact.

Step 1: initial set up of the model

- Set up two parts, one is called *Cylinder* and the other can be called *Plate*, as shown in Figure 6.4 and Figure 6.5, respectively.
- Go to the *Assembly* module, then click on *Create Instance* (as seen highlighted in Figure 6.6), choose the *two parts* that just been defined and bring them together into the assembly module.
- Then click *Translate* (as highlighted in Figure 6.7), move the project to the centre of the plate, and translate it along the z direction.

Step 2: Partition the plate into two regions

- Go back to the *Part* module, then choose *Tools*, *Partition*; this creates a partition window, choose *Cell*, and then click on *Sketch Planar Partition*, as shown in Figure 6.8.
- Click the edge of the Plate part and switch into the *Sketch* module.
- Draw the partition area as shown in Figure 6.9.
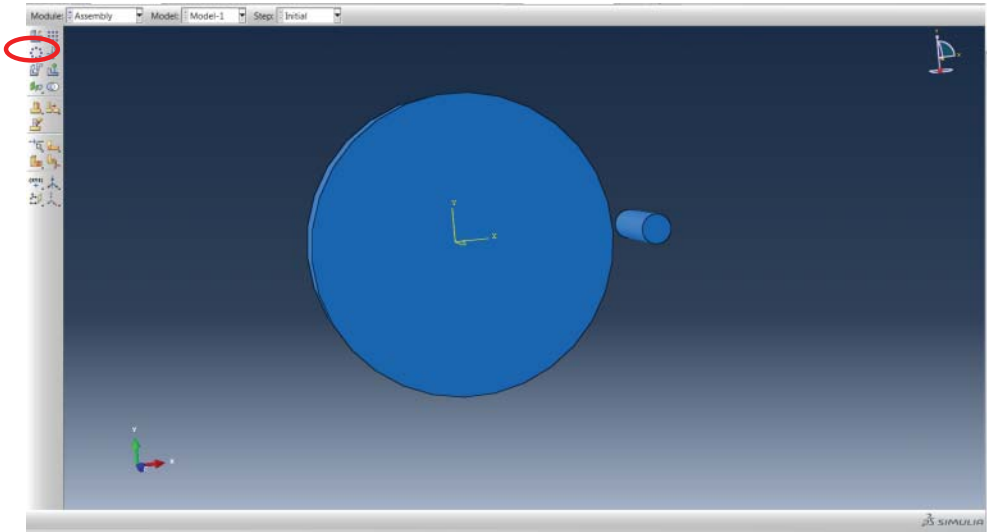
Fig. 6.6   Assembling the two parts.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
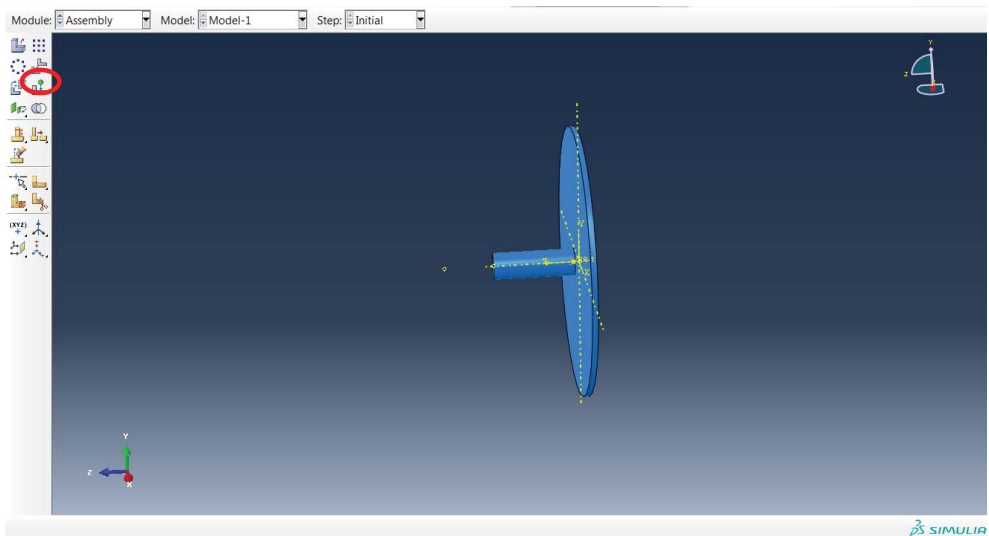
Fig. 6.7   Translate the cylinder to the plate part.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

- Define the data axis by click the *Data Axis* button (as highlighted in Figure 6.10), at the bottom of the screen, choose *X-Axis*, *Y-Axis* and *Z-Axis*.
- Then go back to the *Partition* module, select extrude/sweep edges, and click the circle that has been drawn in the sketch module, as shown in Figure 6.11.
- Then select the data Z-axis as extrusion direction, and create the partition of the plate as shown in Figure 6.12.
- The part has been made into two partitions as shown in Figure 6.13.
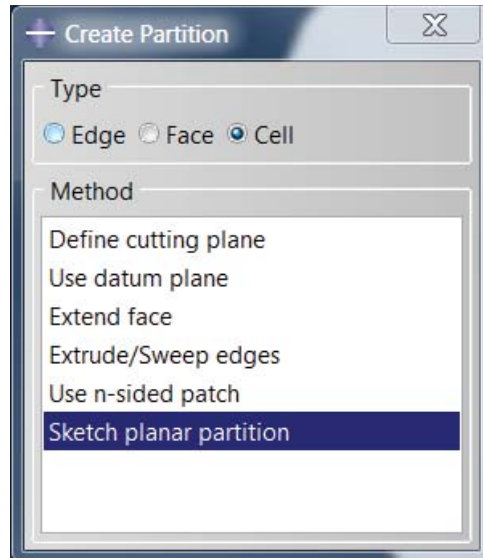
Fig. 6.8   Defining the partition parameter.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
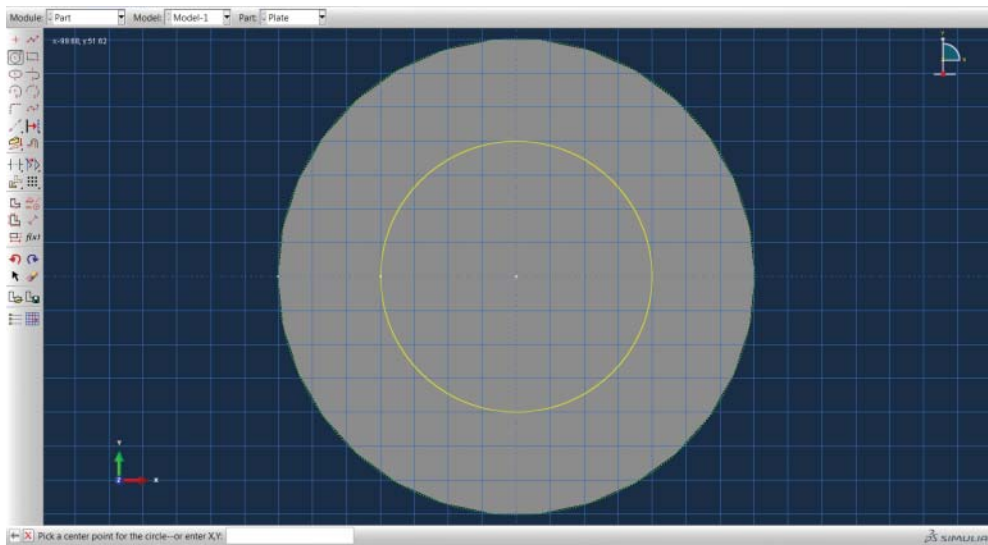
Fig. 6.9   Sketching the partition.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
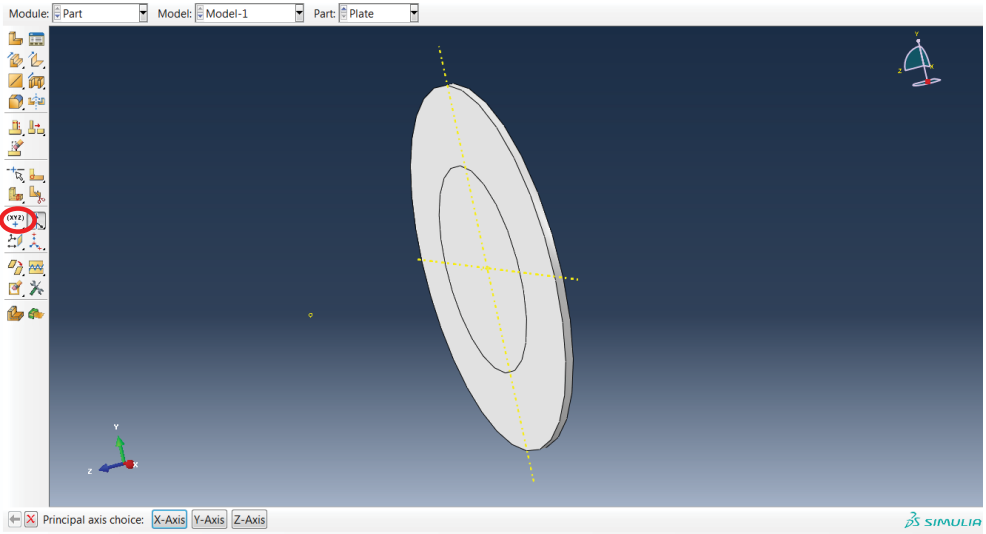
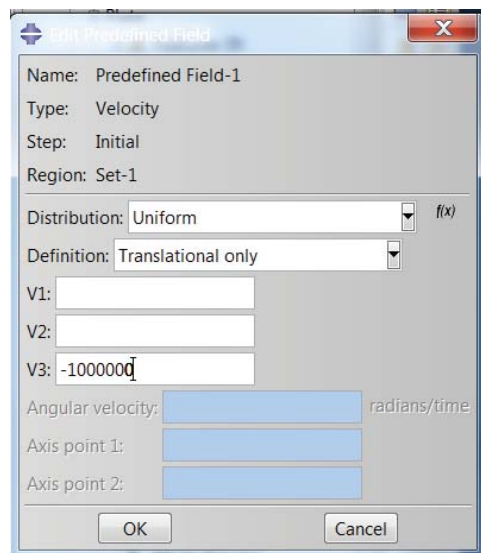**Fig. 6.10   Set up data axes.**
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
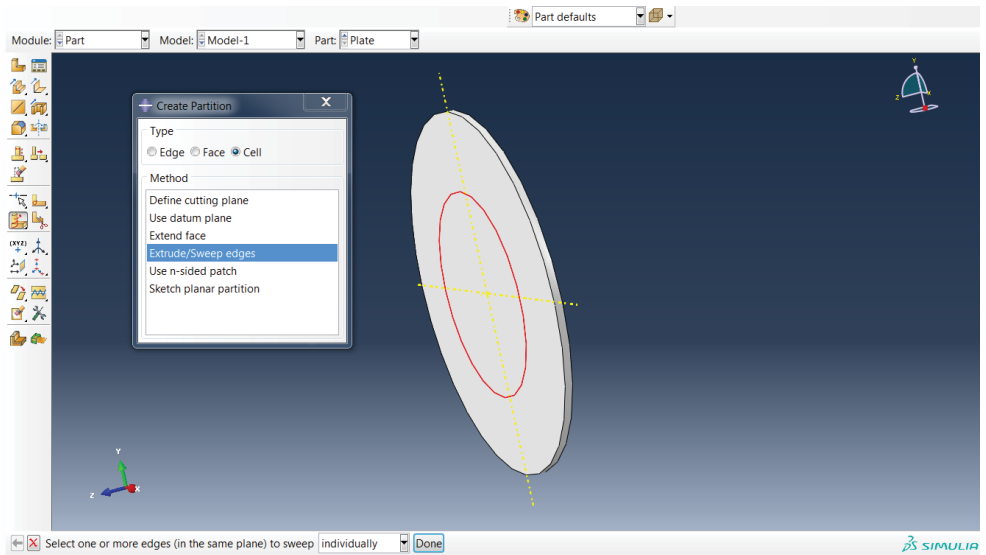
Fig. 6.11   Selecting extrude/sweep edges.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
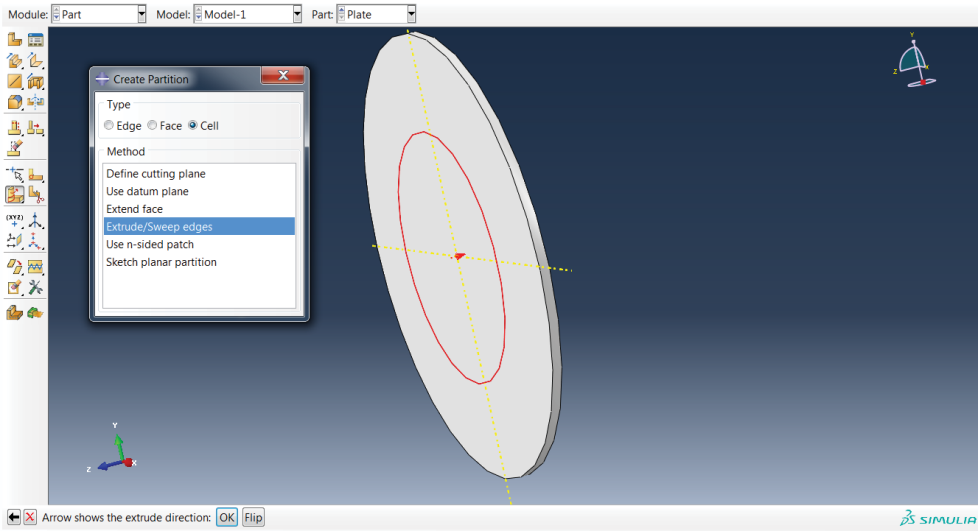
Fig. 6.12   Create partition.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
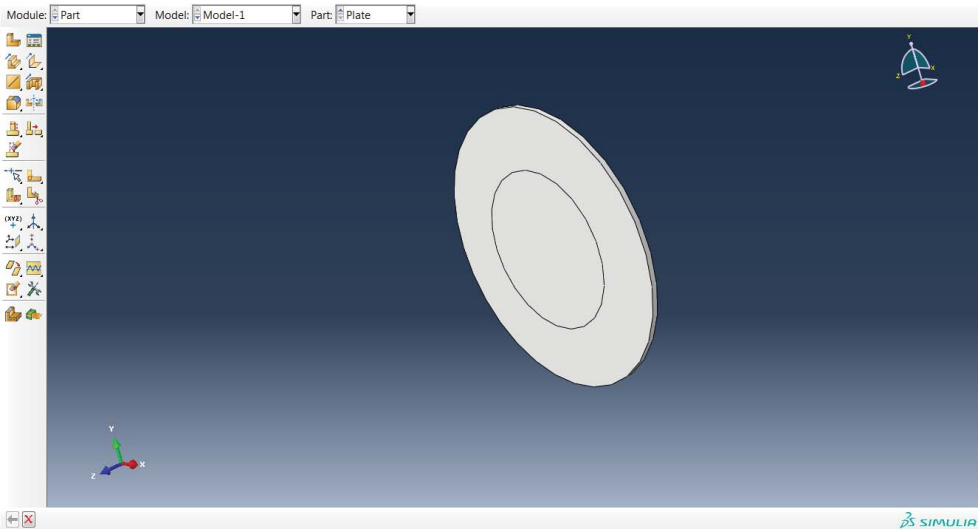
Fig. 6.13   Partition created.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Step 3: Define material

- Go to the *Property* module, then click on *Create Material*, define the *Density*, *Elastic*, *Plastic*, *Shear Damage* and *Ductile Damage* (as shown Figure 6.14).
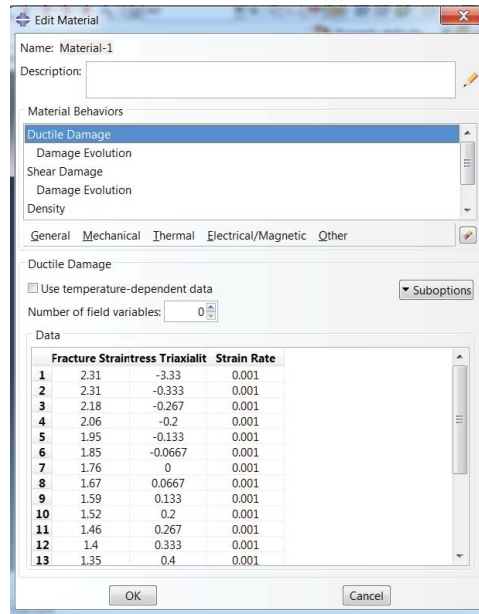
Fig. 6.14   Defining material properties.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Step 4: Define the analysis step

- Go to the *Step* manager module, then click on *Create Step*, choose the *Dynamic, Explicit* procedure as shown in Figure 6.15.
- Then click *Continue*, a new window will pop up as shown in Figure 6.16; make the time period 0.0001.
- In the *Incrementation* tab, make the selection as shown in Figure 6.17.

Step 5: Define the initial velocity of the cylinder

- Go to the *Load* module, click on *Create Predefined Field* in the left column; a new window will pop up (see Figure 6.18).
- Choose *Initial* in the step option, in *Types for Selected Step*, choose *Velocity*, then click *Continue* (see Figure 6.19).
- Then choose the surface of the cylinder (as shown in Figure 6.20), click *OK*, a new window will pop up (Figure 6.21).
- Define the velocity as it is shown in Figure 6.21.
- So the initial velocity of the project has been defined as shown in Figure 6.22.

Step 6: Define self-contact

- Go to the *Interaction* module, click on *Create Interaction Property*.
- A new window will pop up (see Figure 6.23). Choose *Contact*, click *Continue*.
- A new window will pop up (see Figure 6.24). Define the contact property as shown.
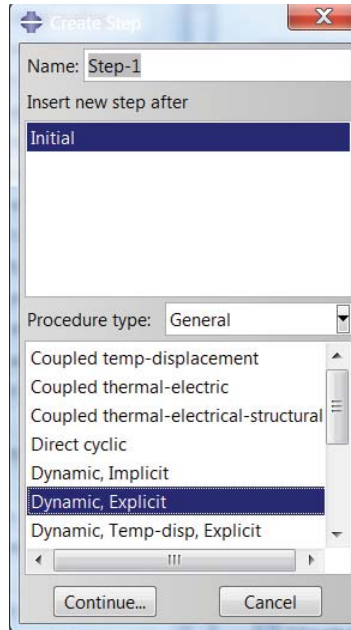
Fig. 6.15   Defining the dynamic explicit step.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
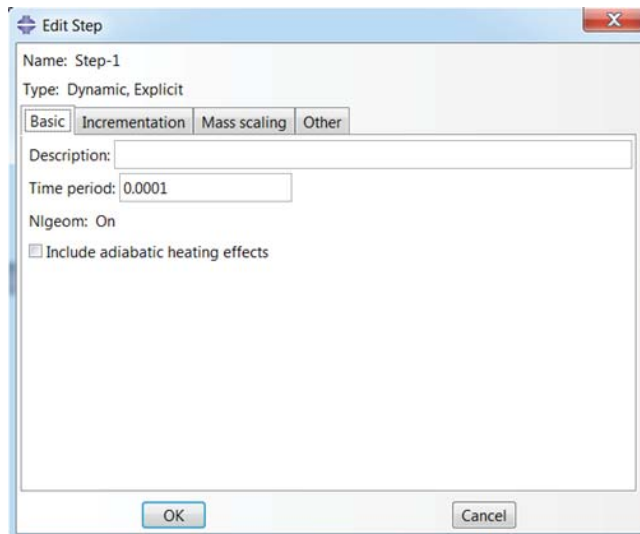
Fig. 6.16   Defining the time period.
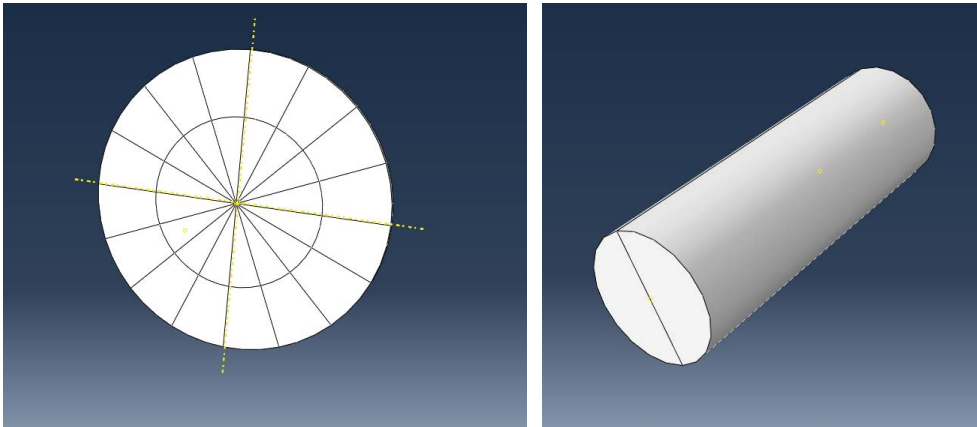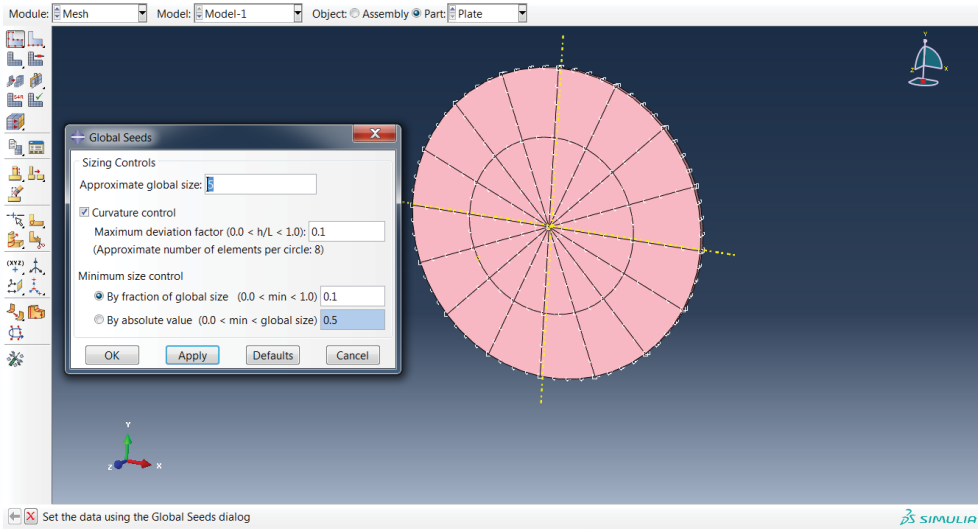**Abaqus® screenshot reprinted with permission from Dassault Systèmes.**
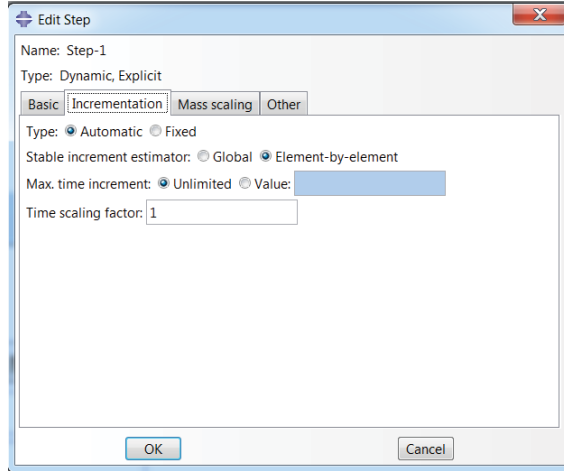
Fig. 6.17   Defining the incrementation.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
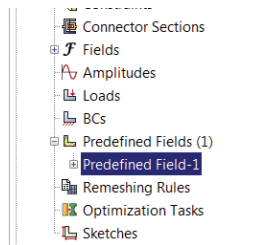
Fig. 6.18   Creating the predefined field.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
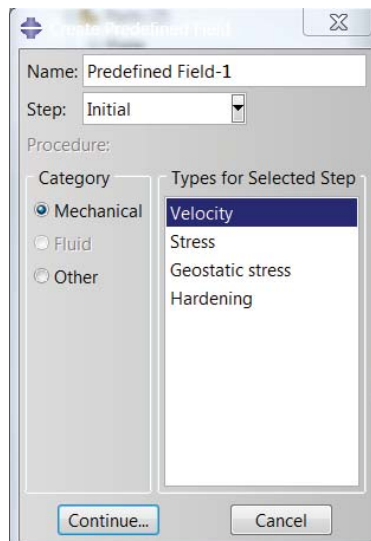
Fig. 6.19   Choosing velocity in a selected step.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
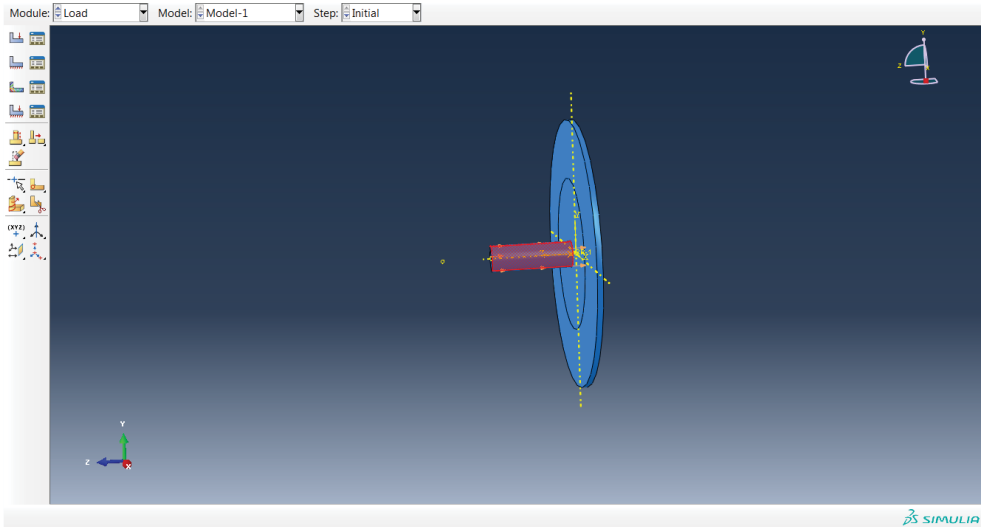
Fig. 6.20   Choosing the surface of the cylinder.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 6.21   Defining the velocity.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

- Go to the *Interaction* module, click on *Create Interaction*, a new window will pop up (see Figure 6.25). Choose *General Contact (Explicit)*, click *Continue*.
- A new window will pop up, define the *General Contact* as shown in Figure 6.26, and choose the interaction '*IntProp1*' we just defined.
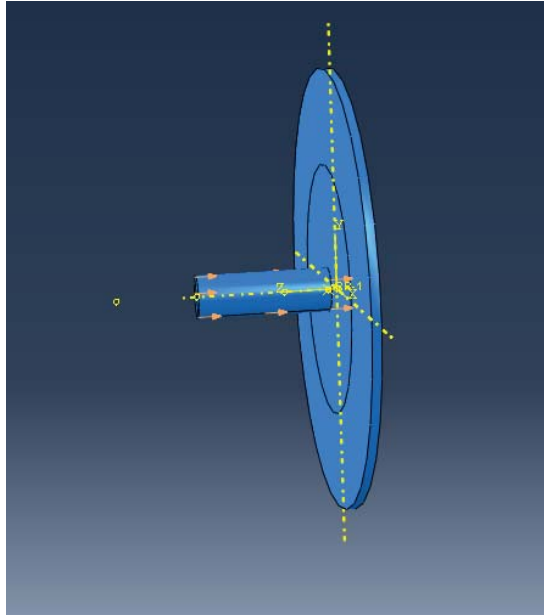
Fig. 6.22 Initial velocity shown in the model.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
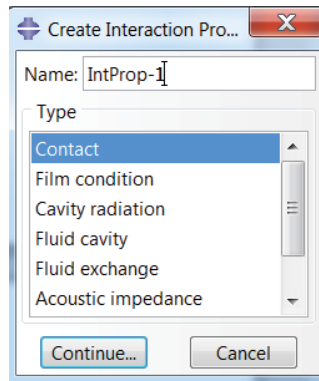
Fig. 6.23 Creating interaction.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Step 7: Mesh definition

- In order to generate a better mesh, a further partition of each part can be defined, as shown in Figure 6.27.
- Go to *Mesh* module, click on *Seed Part*, define the seed (see Figure 6.28).

For the centre of the plate, we will set the mesh to be modelled using SPH when the impact from the cylinder starts to have an effect. It is first modelled using reduced-integration continuum C3D4 elements. A time-based criterion is used to
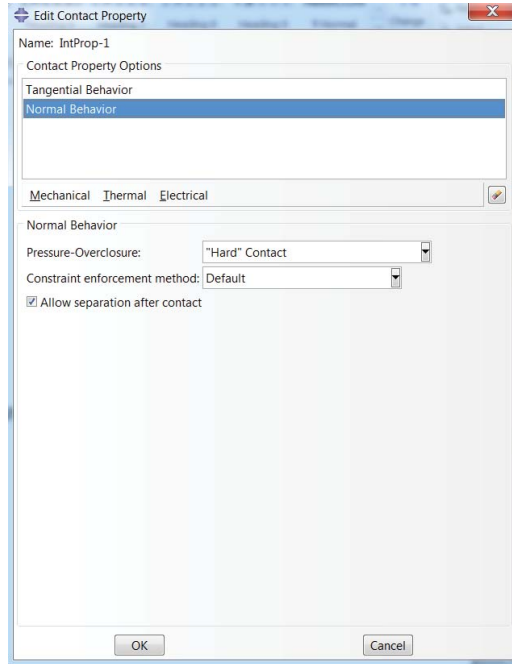
Fig. 6.24   Defining the contact property.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
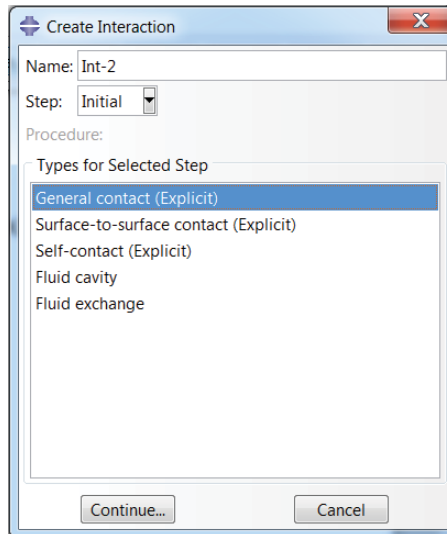
Fig. 6.25   Choosing general contact (explicit).
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
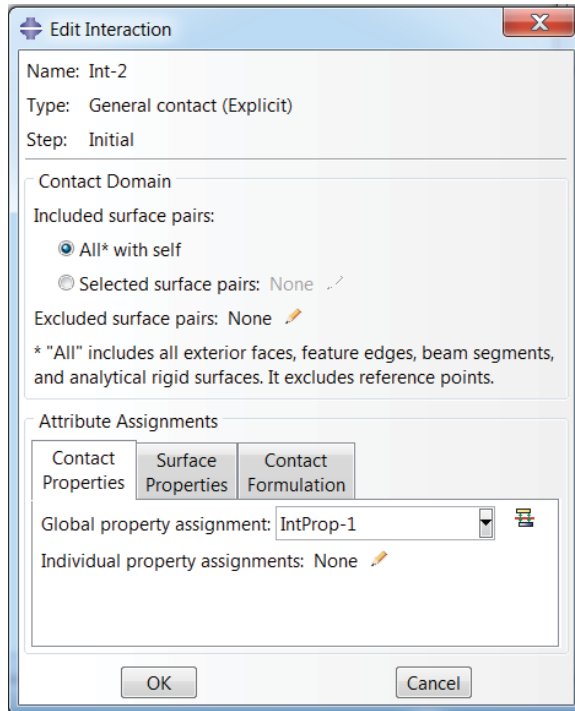
Fig. 6.26   Defining the general contact.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

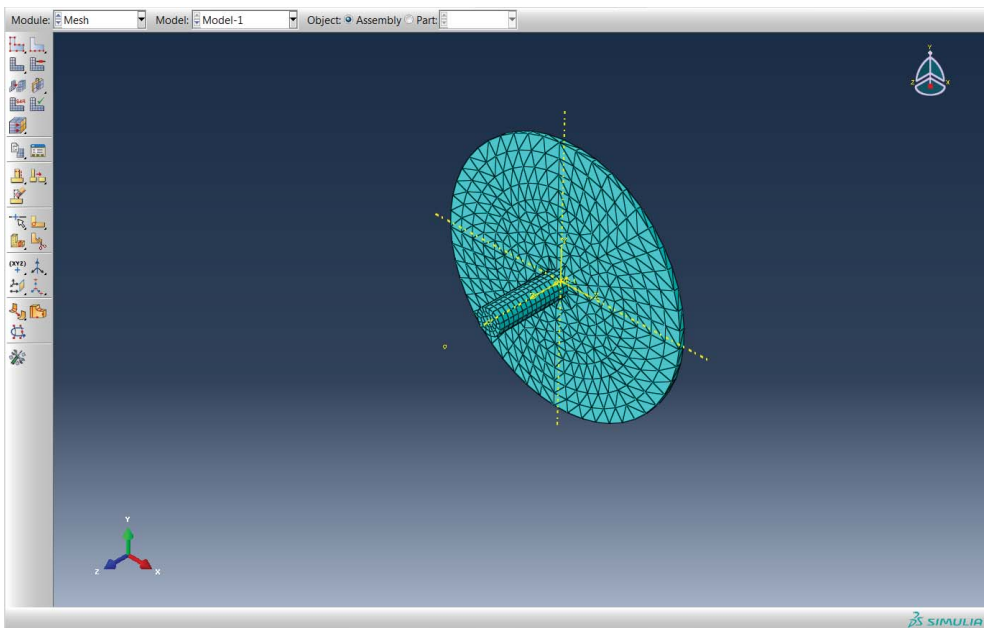**Fig. 6.27   Partition of the part.**
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 6.28   Defining the seed.
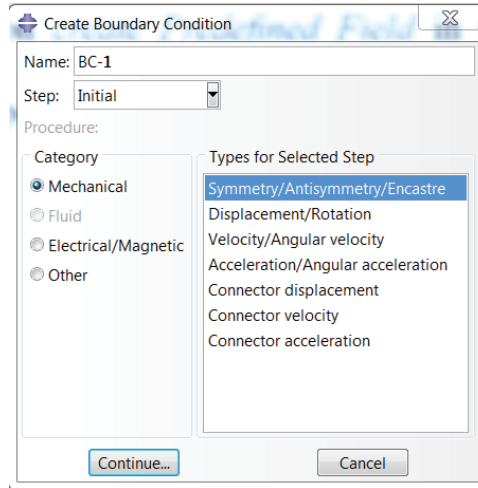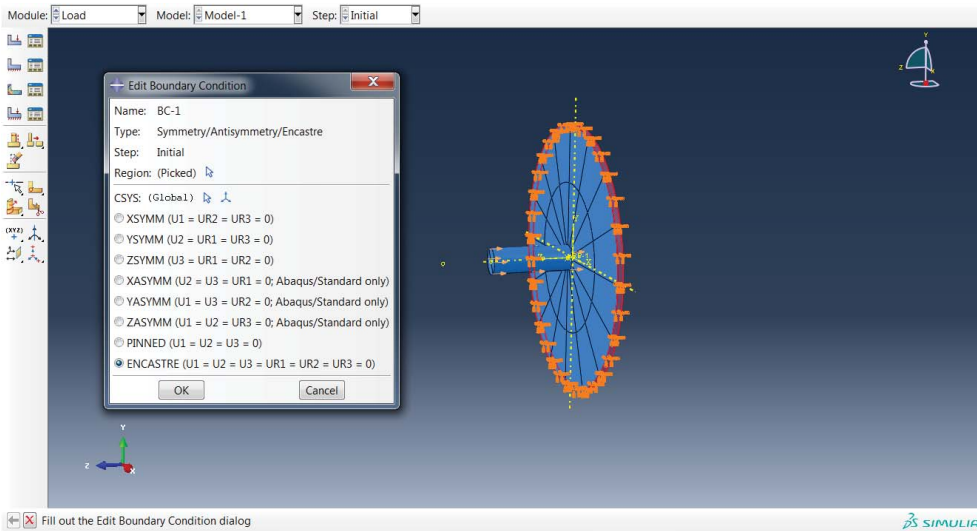Abaqus® screenshot reprinted with permission from Dassault Systèmes.

trigger the conversion from a user-defined mesh to SPH particles (PC3D elements) at the beginning of the analysis. Upon conversion, the continuum elements will become inactive (deleted from the mesh) while the PC3D (SPH particles) become activated.

- Go to *Assign Element Type Mesh*, choose the centre part, click *Done* (see Figure 6.29).
- A new window will pop up as shown in Figure 6.30. Then chose *Element Library – Explicit*, *Family – 3D Stress*, choose *Tet*. In the *Conversion to Particles* option choose yes, in the *Criterion* choose *Time*, *Threshold* choose 0, *PPD* choose 4. This setting means one C3D4 element will convert into 4 SPH particles at time 0.
- Define the element type for the remaining part of the plates as well. However, in *Conversion to Particles* choose *No*; this means they will remain as normal finite elements and conversion to SPH nodes will not be activated.
- The cylinder is modelled with C3D8R elements, as shown in Figure 6.31.
- The final mesh of the structure is shown in Figure 6.32.

The element output available for PC3D elements includes all mechanics-related output for continuum elements: stress; strain; energies; and the values of state, field and user-defined variables. The nodal output includes all output variables generally available in Abaqus®/Explicit analyses.

Step 8: Define boundary condition

- Go to the *Load* module, click on *Create Boundary Condition*, a new window will pop up (see Figure 6.33). Select *Symmetry/Antisymmetry/Encastre*, click *Continue*.
- Define the *Boundary Condition* of the plate, as shown in Figure 6.34.
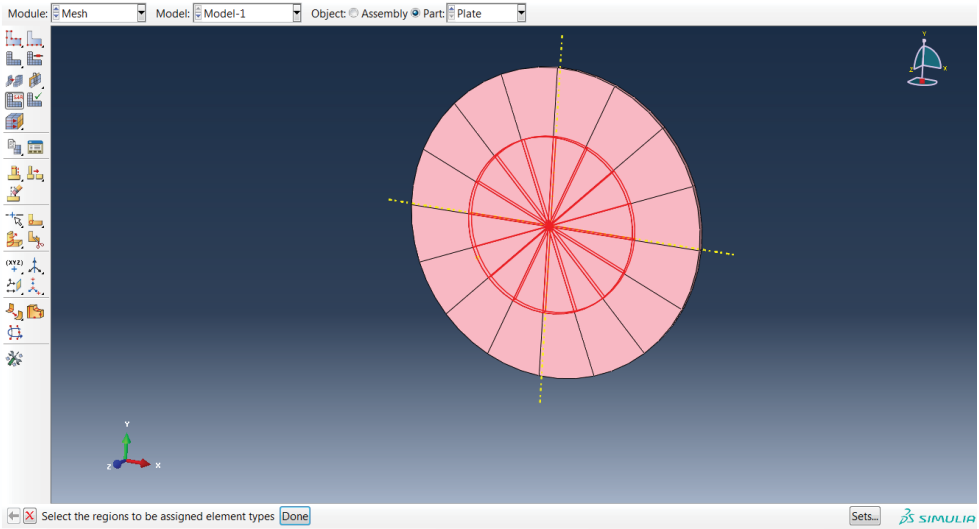
Fig. 6.29   Selecting the centre part.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
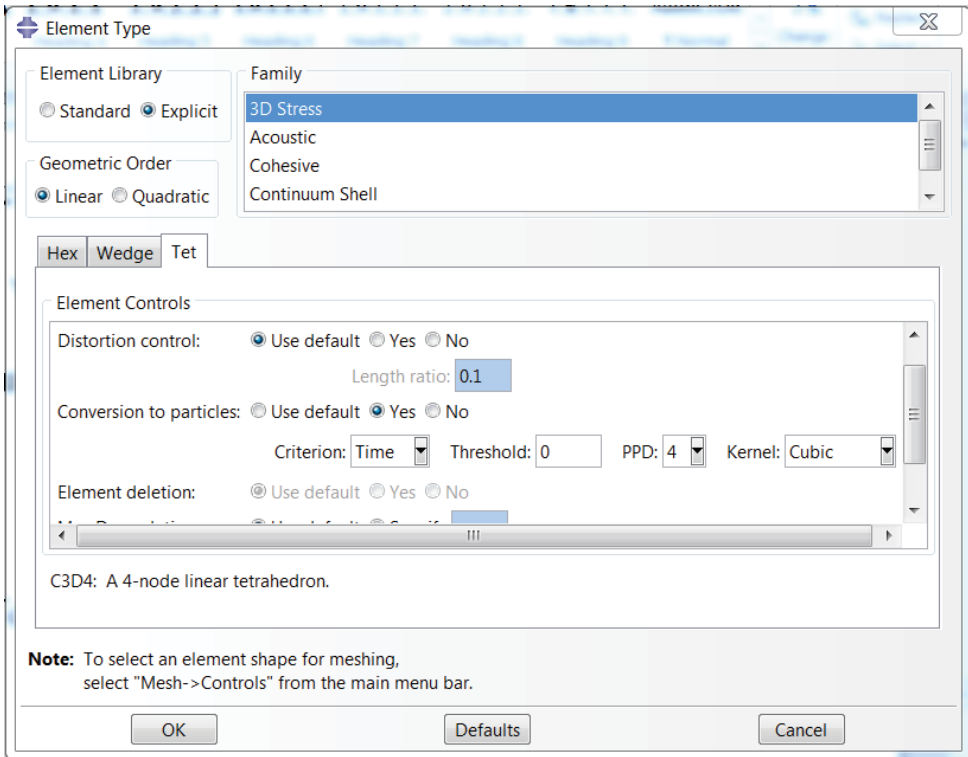
Fig. 6.30   Element definition for the centre area of the plate.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
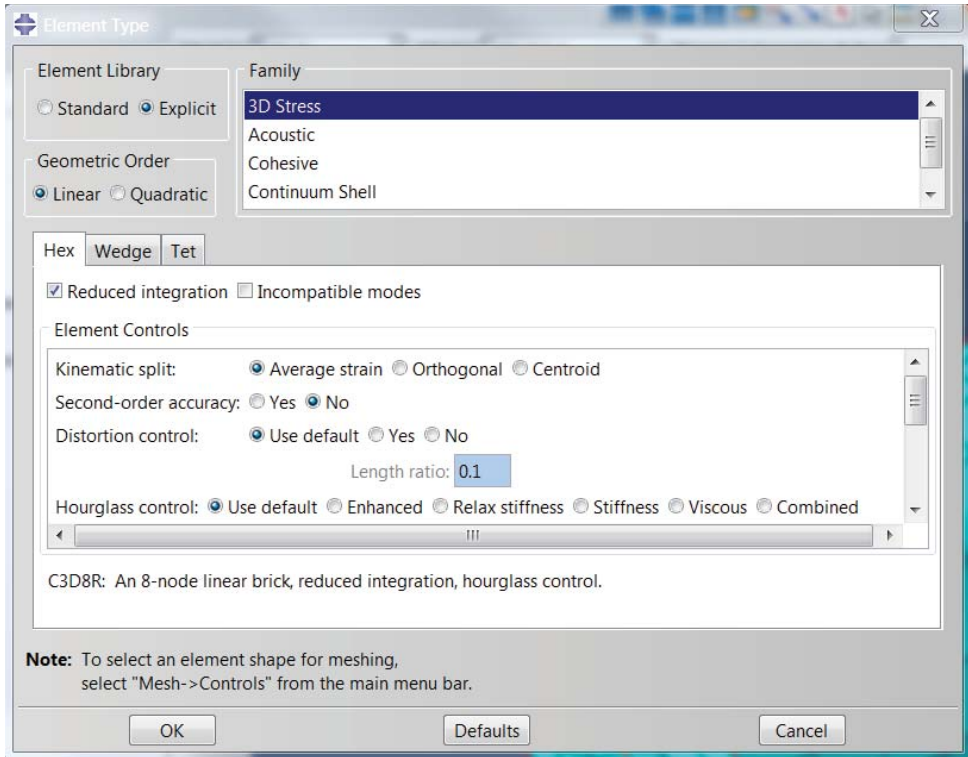
Fig. 6.31   Element definition for the cylinder.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 6.32   Mesh of the structure.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 6.33   Creating the boundary condition.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 6.34   Selecting the boundary condition.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Step 9: Analysis and result interpretation

- Run the analysis; the results can be seen in Figures 6.35 to 6.38. They show the action of the cylinder penetrating the plates and the spread of the debris.

Fig. 6.35   *t*=0.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 6.36   The two parts start to react with each other.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
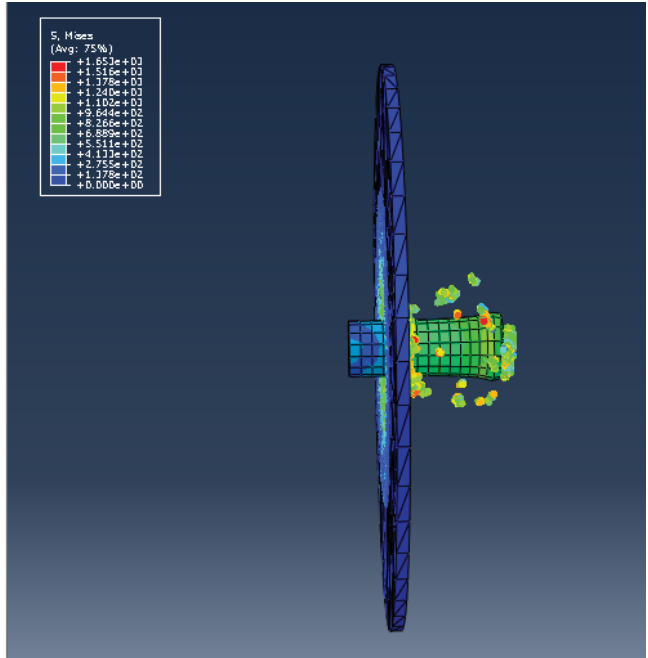
Fig. 6.37 **The cylinder penetrates the plate.**
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
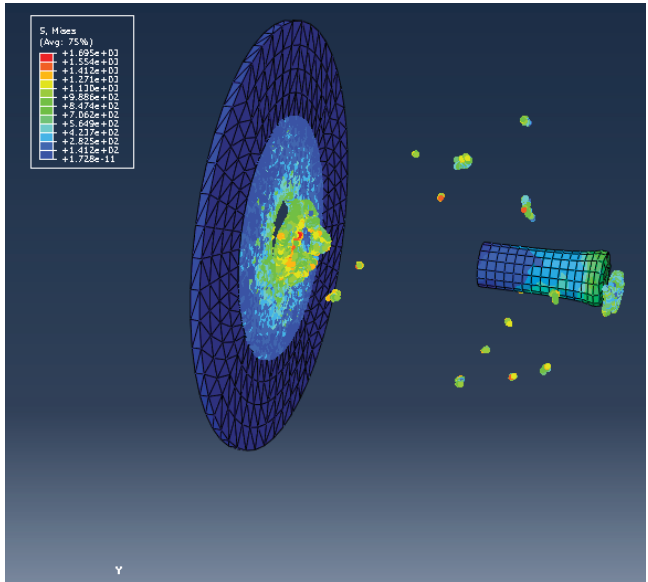
Fig. 6.38 **Final stage of analysis.**
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

References

Astaneh-Asl A. 2003. Progressive collapse prevention in new and existing buildings. Proc., 9th Arab Structural Engineering Conference, Abu Dhabi, UAE, pp. 1001–1008.

Cranz C. 1926. *Lehrbuch der Ballistik*, Springer, Berlin.

Libersky L. D., Petschek, A. G., Carney, T. C., et al. 1993. "High strain Lagrangian hydrodynamics: A three-dimensional SPH code for dynamic material response", *Journal of Computational Physics*, 109. 1, 67-75.

Lucy L. B. 1977. "A numerical approach to the testing of the fission hypothesis," *The Astronomical Journal*, 82, 12: 1013–1024.

Monaghan J. J. 1992. "Smoothed particle hydrodynamics," *Annual Review of Astronomy and Astrophysics*, 30: 543–574.

Yandzio E. and Gough, M. 1999. Protection of Buildings against Explosions. *SCI publication* 244.

Codes and building regulations

FEMA. 2003. Risk Management series, reference manual to mitigate potential terrorist attacks against buildings, FEMA426, Dec 2003.

Hopkinson, B. 1915. British Ordnance board minutes 13565.

National Technical Information Service. 1990. US Departments of the Army, Navy and Airforce, Technical Manual, Army TM5-1300, Navy NAVFAC P-397, Air Force AFR 88-22, Structures to resist the effects of accidental explosions, US Department of Commerce, Washington, DC.

Software and manuals

Abaqus® theory manual. 2013. Version 6.13, Hibbitt, Karlsson and Sorensen, Inc. Pawtucket, R.I.

7 Structural fire analysis

7.1 Introduction

Structural design for fire safety of buildings is an important design task for practising structural engineers. Especially after 911, increasing attention has been focused on the behaviour of buildings under extreme load conditions. The collapse of the World Trade Centre building 7 (WTC7), caused by the fire set by falling debris from the World Trade Centre, drew great attention from engineers and researchers on the study of the structural behaviour of buildings under fire conditions and effective design methods to prevent or delay the collapse of the building. Although in some large consulting companies there is a separate structural fire design group and fire engineers with special training are responsible for fire safety design, for a practising structural engineer or student in structural engineering it is worthwhile getting basic knowledge of structural fire design and knowing a simple and accurate way to model the building under fire conditions.

In current design practice, various fire models and heat transfer analysis methods have been developed. These methods will be briefly introduced in this chapter. The basis for modelling the large deflection and plasticity using an appropriate stress–strain relationship at elevated temperature will also be introduced. Finally, numerical simulation case examples will be introduced.

7.2 Basic knowledge of heat transfer

It is worth introducing some basic heat transfer principles here for a better understanding of the behaviour of a building on fire.

(a) Conduction
Conduction is the mechanism of heat transfer in solid materials in the steady-state situation; the transfer of heat by conduction can be calculated as follows:

$$q = kdT/dx \qquad (7.1)$$

Where:
q is the heat flow per unit (W/m^2)
k is the thermal conductivity

T is the temperature

x is the distance in the direction of heat flow

(b) Convection

Convection is heat transfer through the movement of fluids, either gases or liquids; convective heat transfer is an important factor in flame spread and the transport of smoke and hot gases. It can be calculated as follows:

$$q = h\Delta T \tag{7.2}$$

Where:

h is the convective heat transfer coefficient (W/m^2K)

ΔT is the temperature difference between the surface of the solid and the fluid

(c) Radiation

Radiation is the transfer of energy by electromagnetic waves. It is extremely important in fire as it is the main mechanism for heat transfer. It can be presented as follows:

$$q = \varphi\varepsilon\sigma(T_e^4 - T_r^4) \tag{7.3}$$

Where:

φ is the configuration factor

ε is the Stefan–Boltzmann constant (5.67×10^{-8} W/m^2K^4)

T_e is the absolute temperature of yje emitting surface (K)

T_r is the absolute temperature of the receiving surface (K)

(d) Specific heat

The specific heat is the amount of heat per unit mass required to raise the temperature by one degree Celsius. The relationship between heat and temperature change is usually expressed in the form shown below. The relationship does not apply if a phase change is encountered, because the heat added or removed during a phase change does not change the temperature.

$$Q = cm\Delta T \tag{7.4}$$

Where:

Q is the heat added

C is the specific heat

m is the mass

ΔT is the change in temperature

7.3   Fire development process

Figure 7.1 is a typical time–temperature curve for a conventional process of fire development in a typical room. It can be seen that the curve can be divided into five stages: the growth phase, flashover phase, full developed phase, decay phase and final extinction. However, if a room has very large window openings, too much heat may flow out of the windows for flashover to occur.
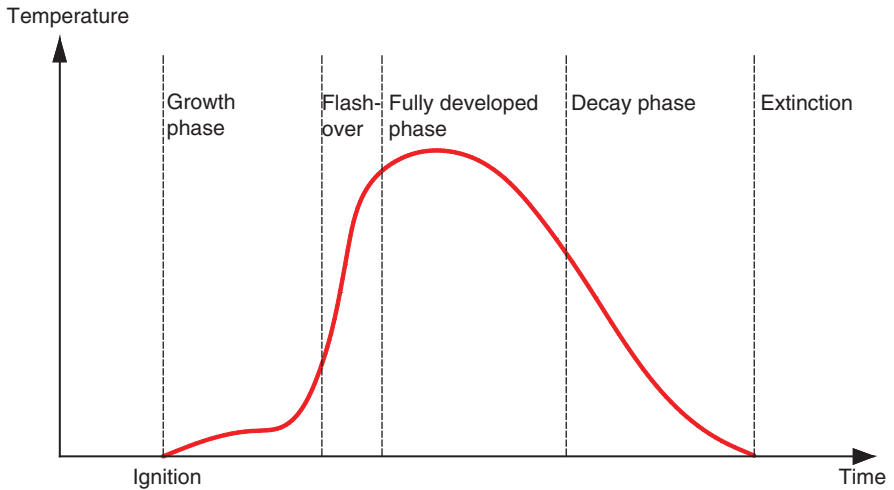
Fig. 7.1   Temperature–time curve for an enclosed fire.
AutoCAD.

(a) Flashover fire
    If the fire is allowed to grow without intervention, and there is sufficient fuel in
    the burning item, temperatures in the burning item will increase radiant heat flux
    to all objects in the room. At a certain level, all combustible items inside the room
    will start to burn, leading to sharp increase in the temperature. This transition is
    called flashover. It is the transition from localised fire to fire of the whole room.

(b) Post-flashover fires
    After flashover, the fire is often referred to as a post-flashover fire, and the rate
    of the combustion normally depends on the size and shape of the ventilation
    openings. However, in a large ventilated room with the fuel items having lim-
    ited combustible surfaces, this is similar to fire in the open air and so becomes
    fuel controlled burning.

(c) Fully developed fire
    A fully developed fire is defined as the stage at which all the available fuel
    within the compartment is burning. Either the available ventilation, or the
    quantity and nature of the fuel, will control the maximum heat release of the
    fully developed fire.

7.4   Fire protection method

There are two main types of fire protection methods, one is active system control the
other is passive system control.

7.4.1   Active system control

Protection is through some action taken by a person or an automatic device, such as
a sprinkler (Figure 7.2) or fire fighters, or control of smoke.
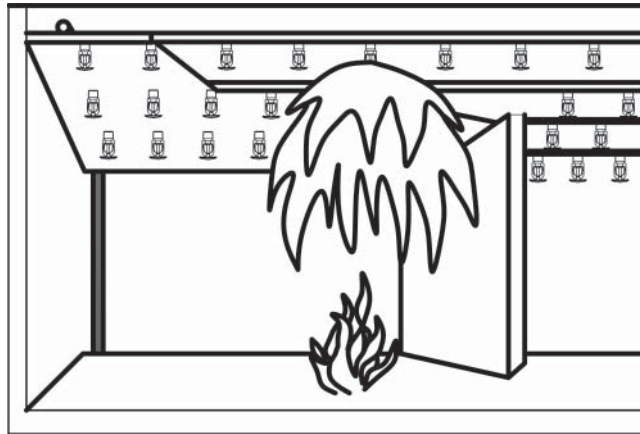
Fig. 7.2 **Sprinkler systems**.
AutoCAD.

7.4.2 *Passive system control*

The fire protection system is built into the structure of the building, such as intumescent paint, spray or board protection on the structural steel members.

Intumescent fire protection is one typically passive fire protection method; one paints a substance on the structural steel members that swells as a result of heat exposure, thus increasing in volume and decreasing in density. This painting is normally finished offsite rather than in the construction site.

Spray fire protection is to spray certain fire resistive materials on the structural members onsite. The thickness of the spray-applied fire resistive material required is determined by testing.

Board fire protection is the use of either gypsum board wrappings or insulation board enclosures on the structural members.

7.5 Fire temperature curve

In real design practice, a fire can be characterised by a time–temperature curve for the atmosphere in the immediate vicinity of the main structural elements. There are two main fire temperature curves used in the practice, which engineers can use directly in their structural fire analysis. They are as follows:

(a) Standard fire temperature curve
   Standard heating by BS476: part 20 (2001) shown in Figure 7.3.
(b) Parametric temperature–time curves[1]

---

[1] Permission to reproduce and derive extracts from British and ISO standards is granted by BSI. British Standards can be obtained in PDF or hard copy formats from the BSI online shop: www.bsigroup.com/Shop or by contacting BSI Customer Services for hardcopies only: Tel: +44 (0)20 8996 9001, Email: cservices@bsigroup.com.
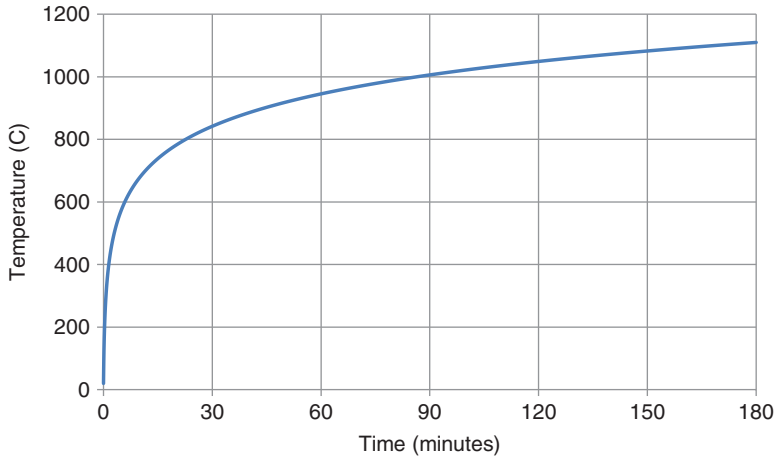
Fig. 7.3   Standard fire temperature curve from BS476: part 20.
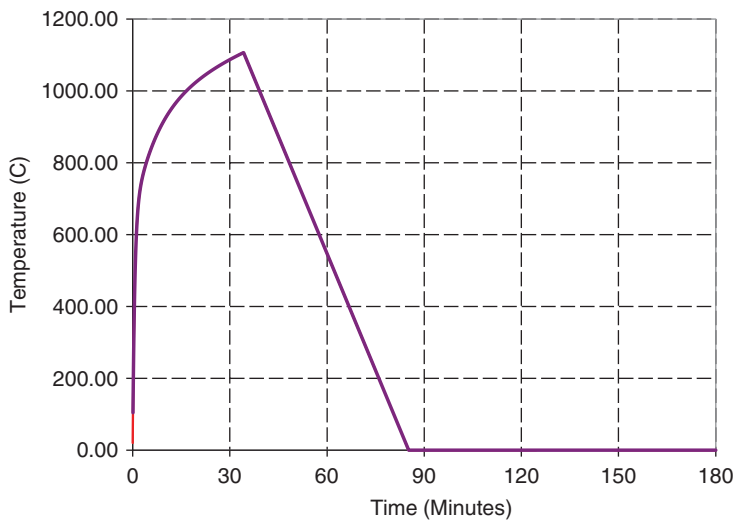
Fig. 7.4   Parametric temperature–time curves from EC1 (EN1991-1-2:2002). Permission to reproduce this extract granted by BSI.

A parametric fire is defined in Annex A of Eurocode 1; Part 1.2 (2002), shown in Figure 7.4. In determining the worst case conditions for the compartment, the main variable is the available percentage of openings.

The temperature rise in this parametric fire differs depending upon: the internal surface area of the room $[A_t]$ $[m^2]$ enclosure including [walls, ceiling and floor, including openings], and the thermal properties [b] $[J/m^2 s^{0,5} K]$ of the wall and ceiling linings. Studies carried out by researchers show that the Eurocode 1 (EN1991-1-2:2002) equation provides the most onerous heating regime for most combinations of fire load and opening factor combinations compared to other methods; therefore, it is widely used in the fire simulation process.

7.6   Determination of the thermal response of structural members

When the atmosphere temperature (also called gas temperature) is determined in the last section, data on the performance of structural elements and fire-resisting can be obtained from tests such as those given in BS 476: Parts 20–22 (2011) and ISO 834-1 (1999), which utilise a standardized temperature–time curve. With these data, we can now determine the thermal response of each structural member, such as the steel beams, columns and slabs. The methodology used in determining thermal response is dependent on the nature of the fire temperature curve as well as the nature of the structural element itself.

Aside from this, Eurocode 3 (2005) gives the formula to work out the increase in temperature for both internal unprotected and protected steelwork. Designers can use it to work out the thermal response of the structural steel members.

Apart from above method, a heat transfer method based upon the finite element method can be used. This will be introduced in the case study section.

7.7   Structural fire design

There are two types of design codes for fire safety in buildings, they can be either of prescriptive type or performance-based type. A prescriptive code states how a building is to be constructed. A performance-based code states how a building is to perform under a wide range of conditions (Custer and Meacham, 1997) which allows the designers to use the fire safety strategy they wish, provided adequate safety can be demonstrated. It allows designers to use fire engineering methods to assess the fire safety of the structure. The key feature for implementing performance-based fire design codes is to assess the fire resistance of the structure. This requires an accurate fire numerical simulation method.

There are major structural fire design codes, such as Eurocode 3, Part 1-2 (2005), Eurocode 4, Part 1-2 (2005) and BS5950 Part 8 (1990) and Eurocode 2 Part 1-2 (2004) to which readers can refer in their design.

The main objective for structural fire design is to make sure that in the event of an outbreak of fire, the load-bearing capacity of the building will continue to function until all occupants have escaped, or been assisted to escape, from the building.

7.7.1   Fire safety design objectives

The main objectives of fire safety design can be summarised as follows:

1. To prevent internal spread of fire.
    A building can be divided into fire compartments using floors or walls or fire doors to prevent the fire spreading for the design fire time, which will increase the time for escape and limit the damage to the structure, thus reducing the loss.
2. To prevent the spread to another building.
3. To prevent the collapse of the building.

7.7.2   Fire safety design framework

The Technical Standard clause 2.0.6 directly references BS 7974 (2002) providing a framework for a fire safety engineering approach. The steps to be taken are:

1. Characterisation of the compartment in terms of layout, size, compartment walls, openings, fire loading and so on.
2. Determination of likely fire conditions in that compartment in terms of atmospheric temperature.
3. Determination of the thermal response of key structural elements to the likely fire conditions.
4. Determination of structural response under the thermal response obtained.

7.8   Major modelling techniques for structural fire analysis

To be able to model the process of the fire development, researchers have developed different methods and equations to determine the temperature–time curve during the fire. Three types of method can be used for determining fire conditions:

1. Standardised models, with recognition of fire conditions and occupancy type.
2. Experimental data appropriate to the compartment and occupancy type.
3. Engineering calculations based upon experimentally calibrated methods.

For method 1, there are two models available to designers, one is the Zone model and the other is the CFD model. For method 3, either manual calculation or finite element modelling can be performed with the use of the fire–temperature curves introduced in the earlier section.

7.8.1   Zone model

Zone models are simple computer models that divide the considered fire compartment into separate zones, where the condition in each zone is assumed to be uniform. The models define the temperature of the gases as a function of time by considering the conservation of mass and energy in the fire compartment. Two-zone models are used for pre-flashover fires whereas one zone models are used for post-flashover fires.

7.8.2   CFD model

Computational Fluid Dynamics (CFD) models can analyse fluid flow, heat transfer and associated phenomena by solving the fundamental equations of fluid flow. These equations represent mathematical statements of the conservation laws of physics. The models can provide information at numerous points within the compartment relating to temperature, velocity, toxic content and visibility. Thus it is one of the most accurate ways to model fire development. However, the input requirement for CFD models is very demanding and requires expertise in defining the correct input parameters and assessing the feasibility of the calculated results; it is not that convenient for a structural engineer to use.

7.8.3   *Finite element method using the fire temperature curve*

The easiest way to perform a structural fire analysis is through the finite element modelling method using the fire temperature curve introduced earlier. The structural response of fire can be simulated by including relevant material properties at elevated temperatures. The method will be demonstrated in the next section using Abaqus®.

7.9   Modelling example of heat transfer analysis using Abaqus®

Heat transfer analysis using Abaqus® is quite convenient for acquiring the thermal response of structural elements. Therefore, in this section a heat transfer example of a concrete slab will be modelled.

The slab is simulated here with the finite element mesh shown in Figure 7.5. The relevant temperature-dependent material properties for concrete as outlined in Eurocode 4 (2005) are used in the analysis.

The fire load is applied in the form of the atmosphere fire curve (this is defined as an amplitude in Abaqus®, which will be explained in detail in a later section) determined using the parametric time–temperature curve applied to the external boundary of the model.

Increments of temperature are controlled by restricting the maximum temperature increase in any one step to 10 °C (please also refer to Figure 7.23 for defining the step parameter in Abaqus®). The solution then proceeds by time stepping through the problem for the designated analysis duration.

7.9.1   *Model set up*

1. Click Abaqus® CAE to start Abaqus® and choose to create a new model *Database*, with a standard/explicit model.
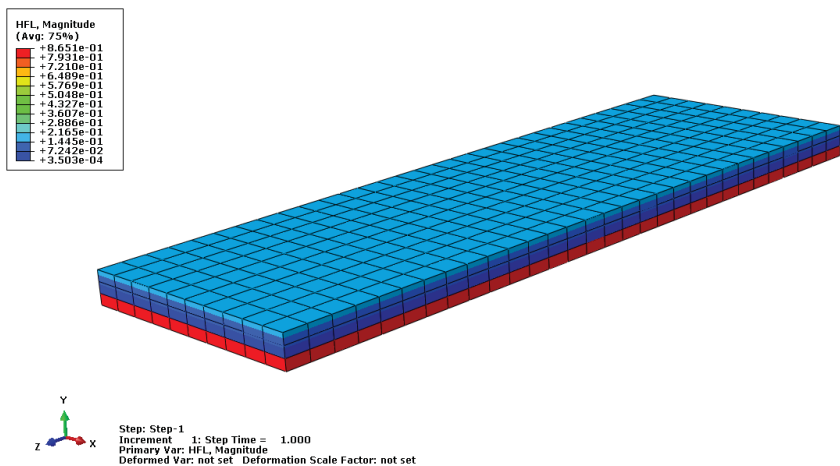
Fig. 7.5   Slab under elevated temperature.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

2. From the window shown in Figure 7.6 select the *Part Module*, click *Parts* which is located on the left hand side, known as the model tree, or click on *Create Part Icon* on the vertical ribbon.

3. A new window will pop up (as shown in Figure 7.7); type the name of the part *Slab*, for *Modelling Space* choose *3D. Type – Deformable*; *Base Feature – Solid*; *Approximate Size – 20* (note: we presume this is in m here, so throughout the remaining modelling all the units, such as strain and stress, should use mm), then click *Continue*.

4. Create the geometry of the slab by drawing directly in the *Sketch*, as shown in the image below (Figure 7.8). You can define each control point either by using the mouse or inputting the x-y coordinate.

5. Click *Done*, a new window will pop up as shown in Figure 7.9; put the *Depth* as 3 m, which is the length of the slab.
   The slab is shown in Figure 7.10.

6. Go to the *Property* module, or double click on the *Materials* tab in the model tree (Figure 7.11).
   - Name the new material 'concrete'.
   - Click on the *Thermal* tab then *Conductivity*; put in the conductivity of the concrete at different temperatures according to Eurocode 4 EN1994-1-2:2005 (see Figure 7.12).
   - Click on the *Thermal* tab then *specific heat*; put in the specific heat value of concrete at different temperatures according to Eurocode 4 EN1994-1-2:2005 (see Figure 7.12).
   - Define the density of the concrete by clicking *General* then *Density* (see Figure 7.12).
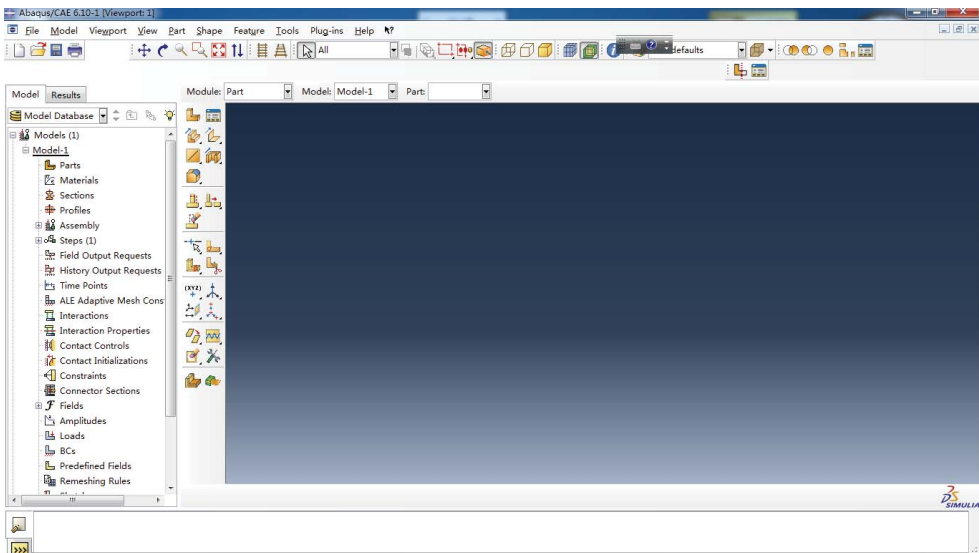
Fig. 7.6   Starting CAE.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
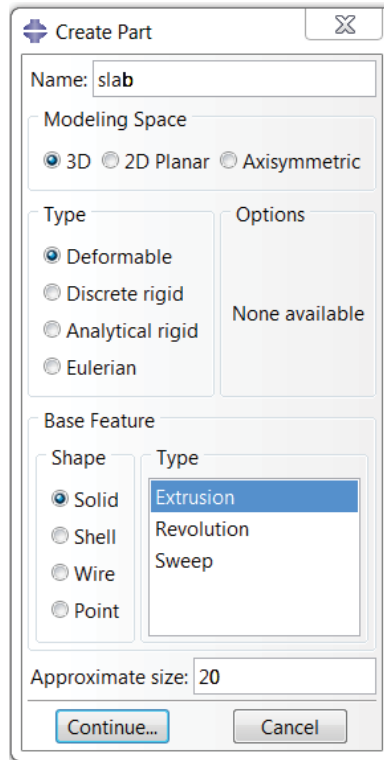
Fig. 7.7   Defining the slab part.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 7.8   Sketching the slab.
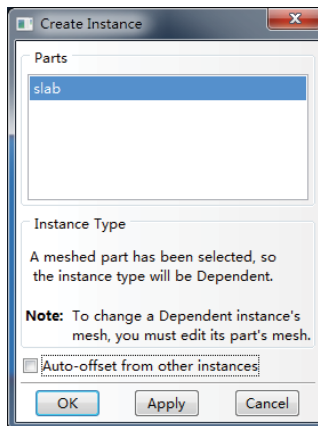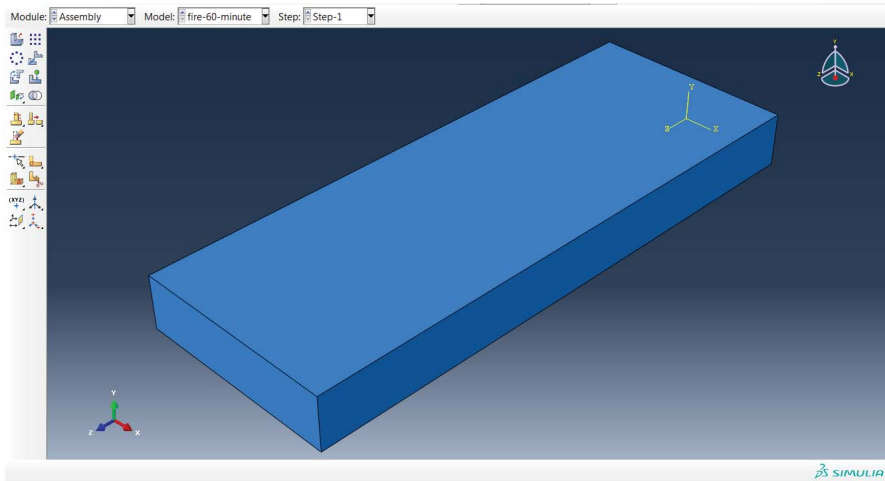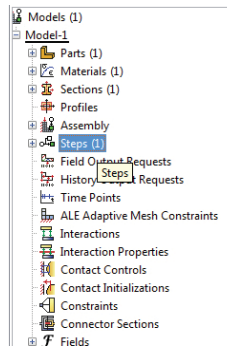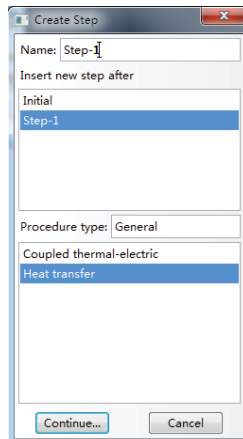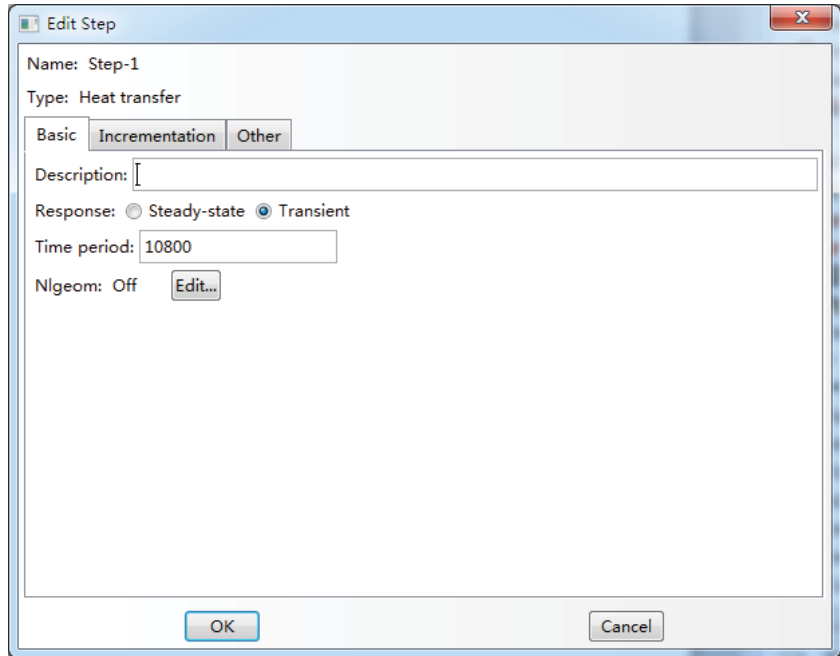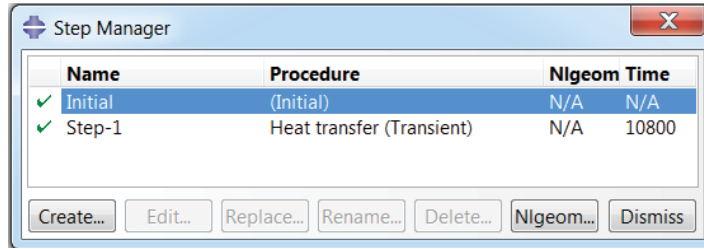Abaqus® screenshot reprinted with permission from Dassault Systèmes.
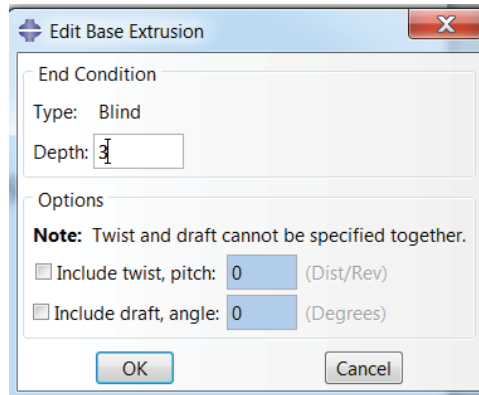
Fig. 7.9   Defining the depth of the slab.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
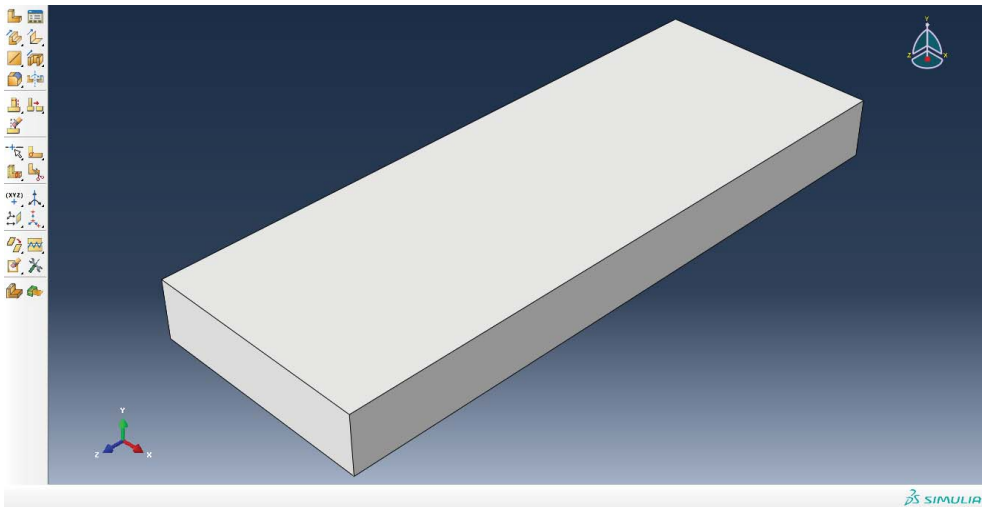
Fig. 7.10   Slab being generated.
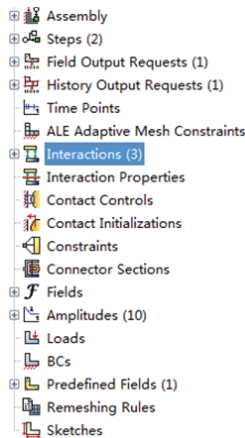Abaqus® screenshot reprinted with permission from Dassault Systèmes.
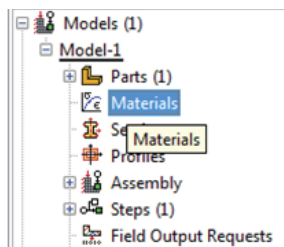
Fig. 7.11   Selection of material module.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
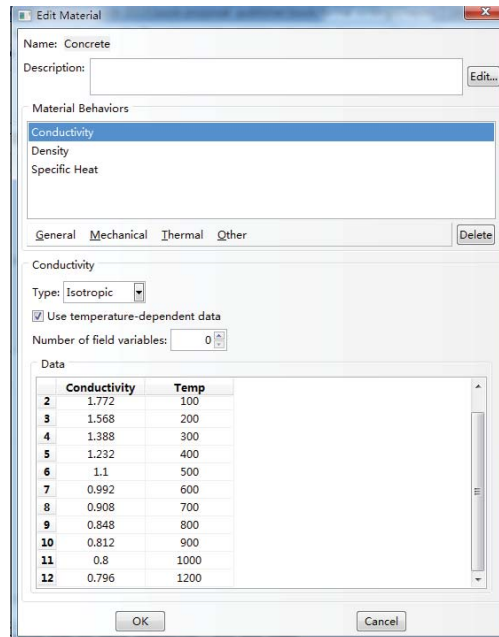
Fig. 7.12   Defining material of concrete.
Abaqus$^{®}$ screenshot reprinted with permission from Dassault Systèmes.

Remember, units have to be kept consistent within Abaqus$^{®}$.
7.   Double click on the *Sections* module in the model tree (Figure 7.13).
   • Name the section 'concrete' and select *Solid* for the category and *Homogeneous* for the type (Figure 7.14).
   • Select the material we have created (*Concrete*) (Figure 7.15).
8.   Expand the *Parts* tab in the model tree and double click on *Section Assignments*.
   • Select the surface geometry in the viewport (it becomes highlighted in red, see Figure 7.16).
   • Then, in the *Edit Section Assignment* tab, select the section created above (*Concrete*), see Figure 7.16.
9.   Expand the *Assembly* tab in the model tree and then double click on *Instances* (Figure 7.17).
   • Select *Dependent* for the instance type.
   • Then select parts *Slab* (Figure 7.18); the slab will be assembled into the assembly module as shown in Figure 7.19.
10.   Double click on *Steps* in the model tree (Figure 7.20).
   • Name the step, set the procedure to *General*, and select *Heat Transfer* (Figure 7.21).
   • Give the step a description 'Step-1'.
   • A new window will pop up as shown in Figure 7.22. In *Time period* put in 10800, which stands for 10800 seconds (it is 3-hour fire duration).
   • In the *Incremental* tab, put the parameters as shown in Figure 7.23; as mentioned earlier, the temperature Increment is 10 °C, so we put 10 here in the *Max allowable temperature change per increment*.
   • The heat transfer step is then defined as shown in Figure 7.24.
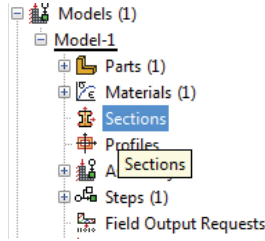
Fig. 7.13   Selecting sections module.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
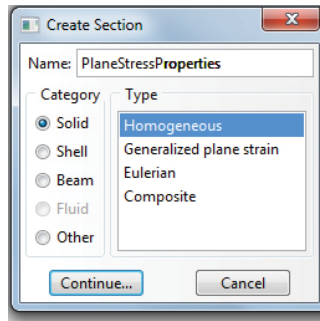
Fig. 7.14   Creating a section.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 7.15   Choosing the concrete material.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

7.9.2   Define the heat transferring parameters

In this example, both convection and radiation need to be defined in Abaqus® as follows:

Convection is worked out automatically by Abaqus® which calculates the heat flux applied to the model by using a heat convection coefficient of 35 W/m²K. The top surface of the slab is modelled assuming heat loss by convection to room temperature (20 °C) using a heat convection coefficient of 9 W/m²K.

Radiation, assuming a conservative emissivity of 0.8 for concrete, as outlined in PD7974-3 (2003) for a natural fire.
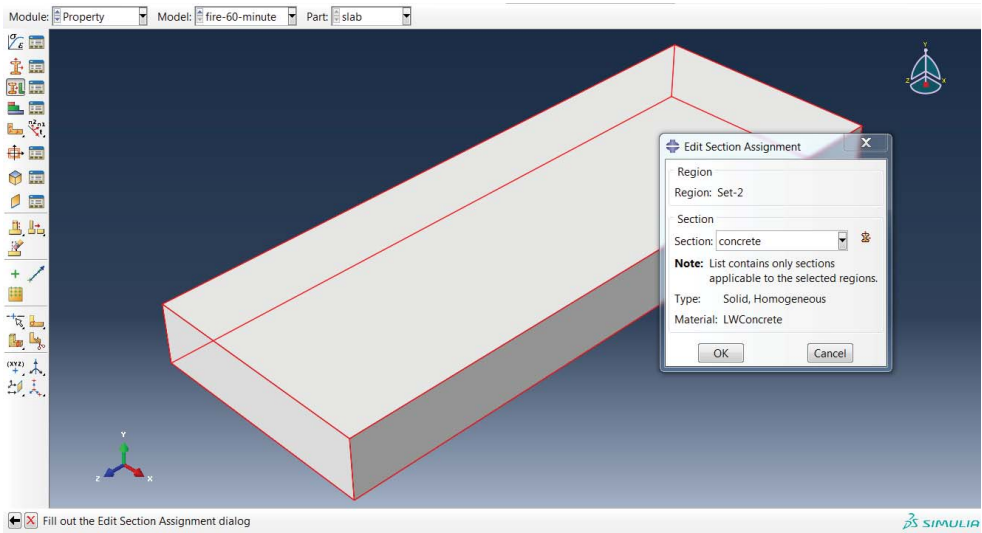
Fig. 7.16   Assigning concrete material to the slab.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
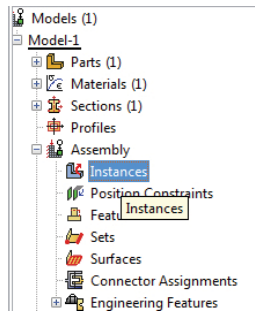
Fig. 7.17   Selecting the instance.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 7.18   Selecting the slab as the instance.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 7.19   Slab being added to the assembly.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 7.20   Choosing the step module.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 7.21   Defining the heat transfer analysis step.
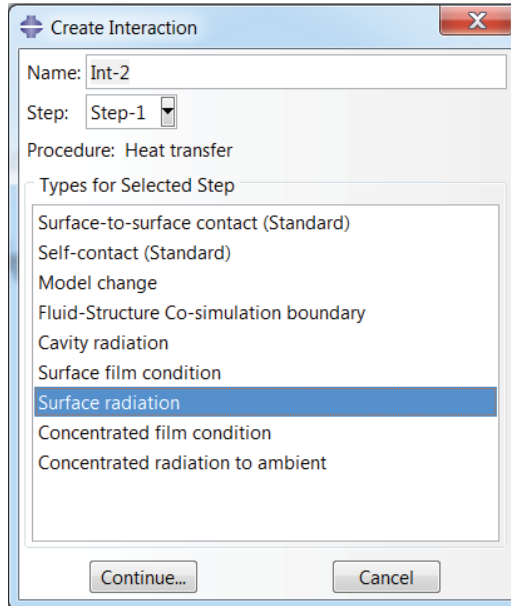Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 7.22   Defining basic parameters in the *Step* module.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
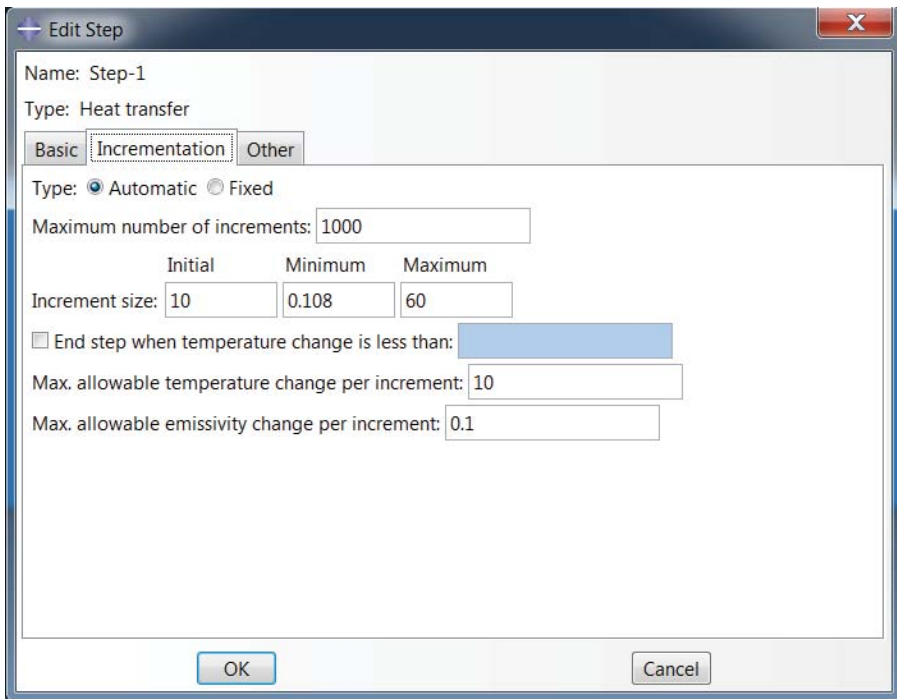
Fig. 7.23   Defining the incrementation.
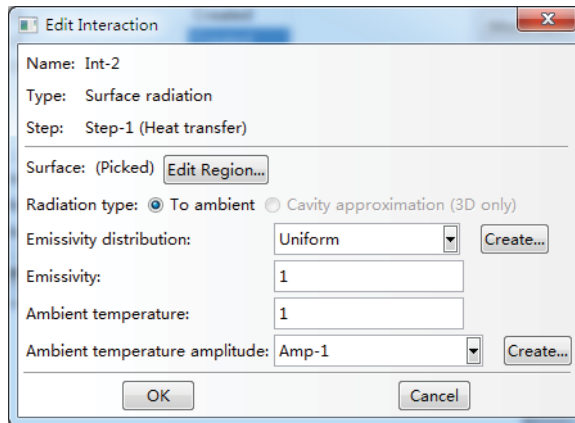Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 7.24 Heat transfer step is defined.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 7.25 Choosing the interaction module.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Define the convection

Here the convection is defined, to simulate the convection from the bottom edge to the top surface.

Define surface film condition for the bottom edge

- Expand *Interaction* in the model tree, and then create interaction (Figure 7.25).
- Name the interaction (*Int-1*); choose the *Surface film condition* (Figure 7.26).
- Click *Continue* and a new window will pop up, then select the surface of this interaction by choosing the bottom edge of the slab (Figure 7.27).
- A new window will pop up (as shown in Figure 7.28), then define the parameters for the *Surface film condition*; make the selection as follows (as shown in Figure 7.28):
  - *Definition* – select *Embedded Coefficient*.
  - *Film coefficient* – 35.
  - *Film coefficient amplitude* – *Instantaneous*.
  - *Sink Temperature* – 1.
  - *Sink Amplitude* – click *Create*.
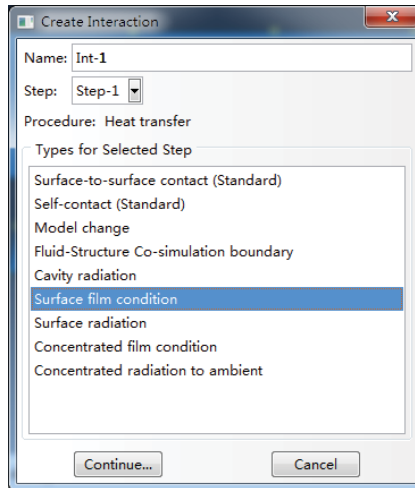
Fig. 7.26   Selecting the surface film condition.
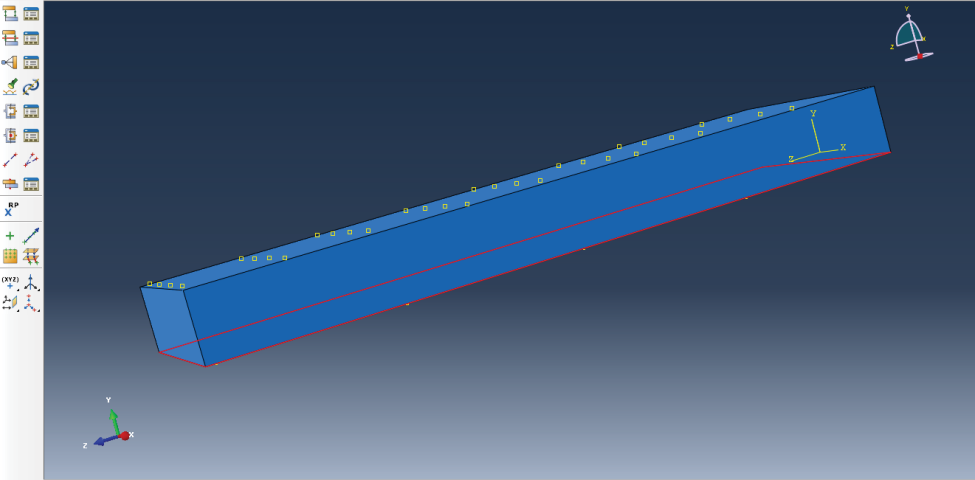Abaqus® screenshot reprinted with permission from Dassault Systèmes.
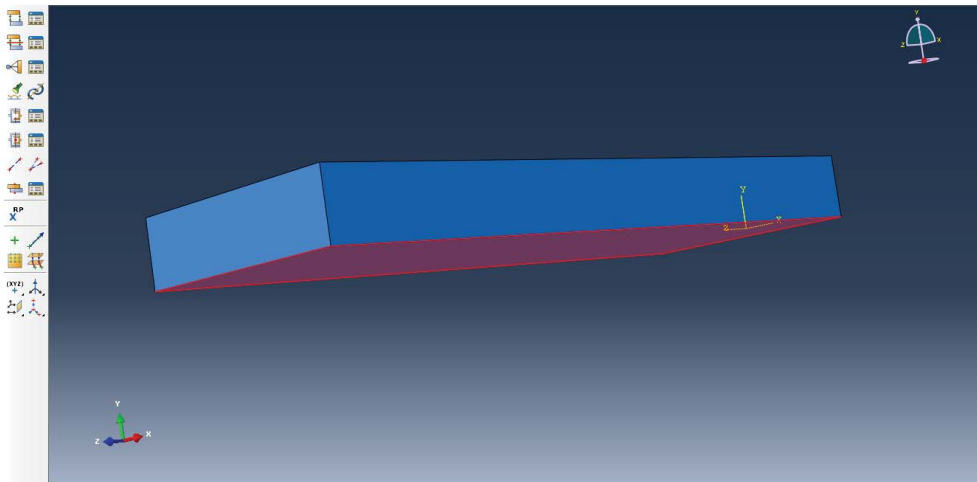
Fig. 7.27   Selecting the bottom surface of the slab.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

- A new window will pop up (Figure 7.29); name the amplitude *Amp-1*, choose *Tabular*.
- A new window will pop up (Figure 7.30).
- Copy and paste the atmosphere fire temperature curve (parametric fire curve) worked out based on Eurocode 1 (2002) as explained in Section 7.5.

Thermal film conditions: explanation

Film condition interactions define heating or cooling due to convection by surrounding fluids. Two types of film condition interaction are available in Abaqus®/CAE: sur-
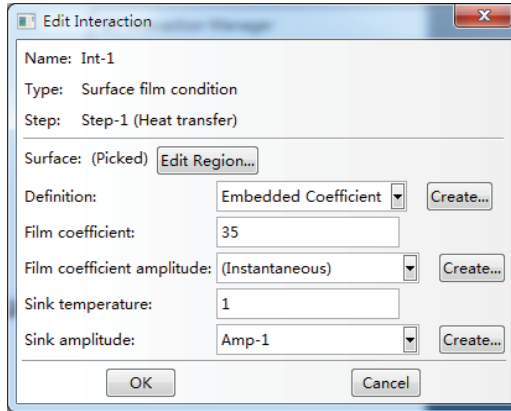
Fig. 7.28   Defining the surface film condition.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
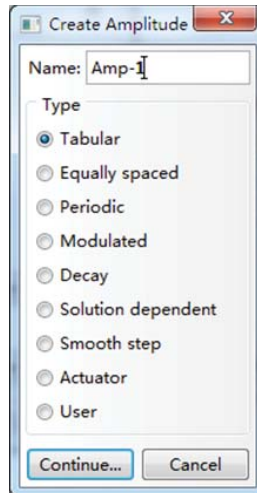
Fig. 7.29   Choosing the fire temperature amplitude.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

face film conditions define convection from model surfaces, and concentrated film conditions define convection from nodes or vertices (Abaqus® 2013).

Define surface film conditions for the top edge

- Expand *Interaction* in the model tree, and then create an interaction, choose surface film condition. Define the parameters as shown in Figure 7.31.
- Then choose the top edge, as in Figure 7.32.

Define the radiation

Here the radiation will be defined in order to simulate the heat radiating from the bottom surface to the top surface.
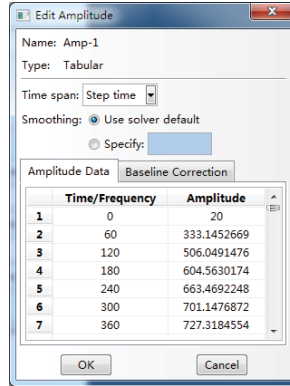
Fig. 7.30   Fire temperature amplitude.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
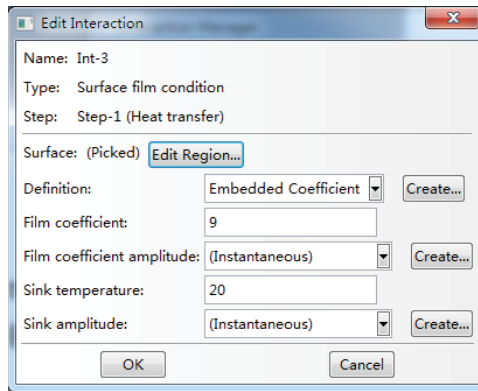
Fig. 7.31   Defining surface film conditions for the top edge.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
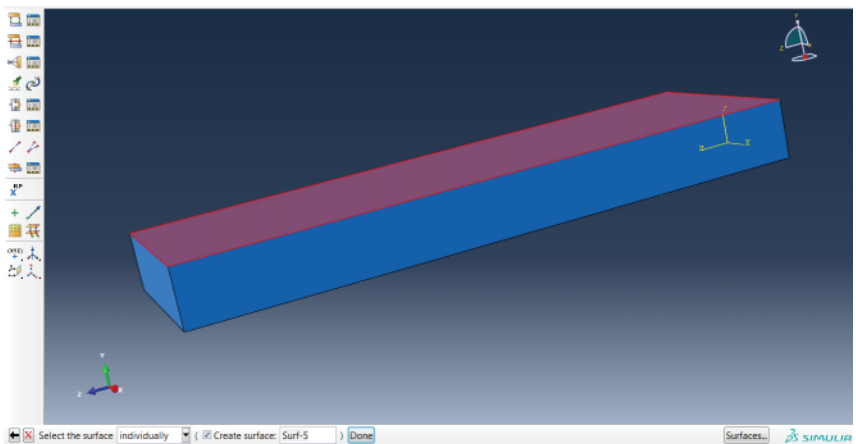
Fig. 7.32   Selecting the top surface of the slab.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 7.33   Creating surface radiation.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 7.34   Defining the surface radiation parameter.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

- Expand *Interaction* in the model tree, and then create an interaction, choose *Surface radiation* (Figure 7.33).
- Define the parameters as shown in Figure 7.34, also choose the bottom edge as shown in Figure 7.35.

Radiation to and from the ambient environment: explanation

Radiation interactions describe heat transfer to a non-reflecting environment due to radiation. Two types of radiation interactions are available in Abaqus®/CAE: surface

Fig. 7.35   Selecting the bottom surface of the slab.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

radiation interactions describe heat transfer with a non-concave surface, and concentrated radiation interactions describe radiation from nodes or vertices.

Defining output

- Expand *Field Output Requests* in the model tree, and then double click on *F-Output-1* (F-Output is automatically generated when creating the step) (Figure 7.36).
- The *Field Output Request* window will pop up as shown in Figure 7.37, set the parameters as shown.
- The output variables that can be chosen in the analysis are HFL (heat flux vector at integration points), NT (nodal temperature at each node) RFL (reaction fluxes at nodes).

You can also define the *History Output Request* as shown in Figure 7.38.
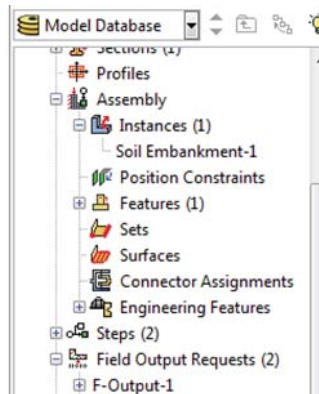
Fig. 7.36   Selecting the field output.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
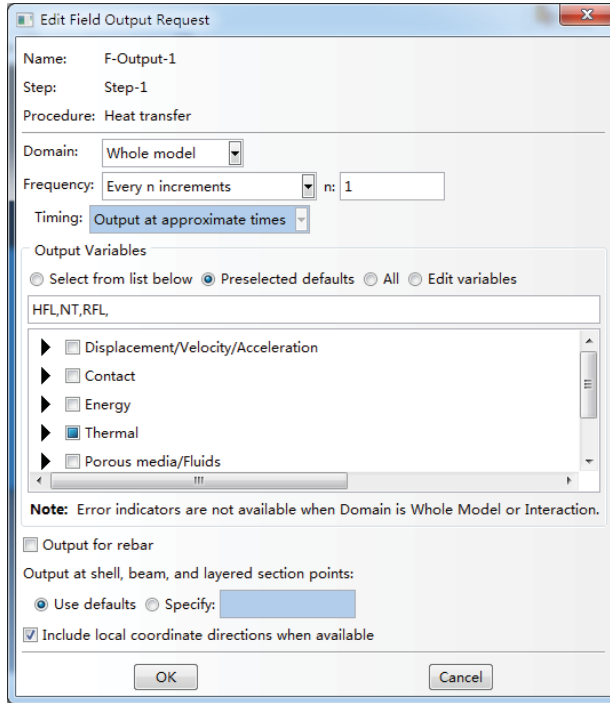
Fig. 7.37   Defining the field output parameter.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
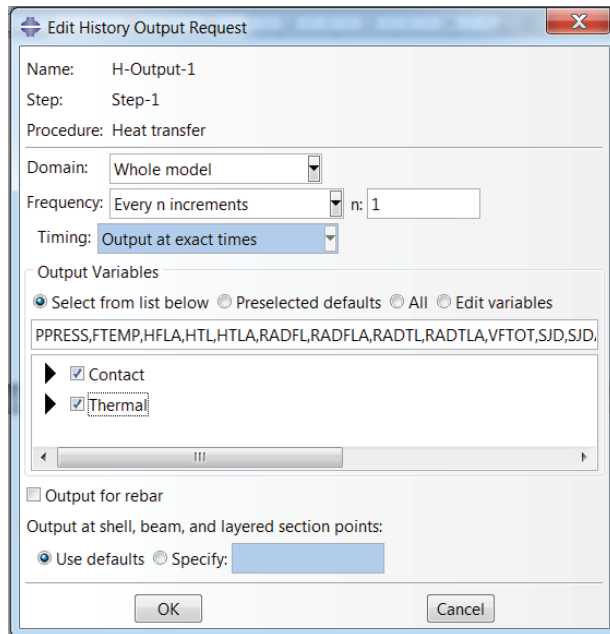
Fig. 7.38   Defining the history output.
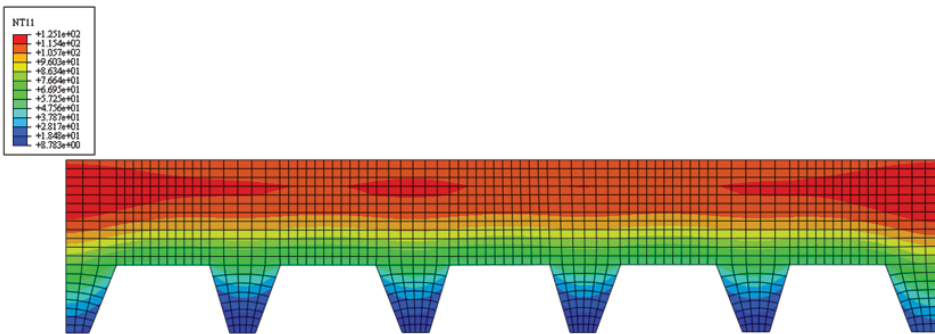**Abaqus® screenshot reprinted with permission from Dassault Systèmes.**
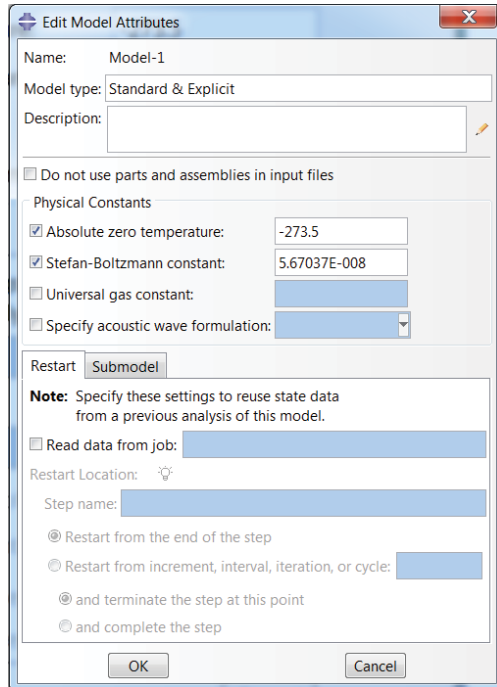
Fig. 7.39 Defining absolute zero temperature and the Stefan–Boltzmann constant.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
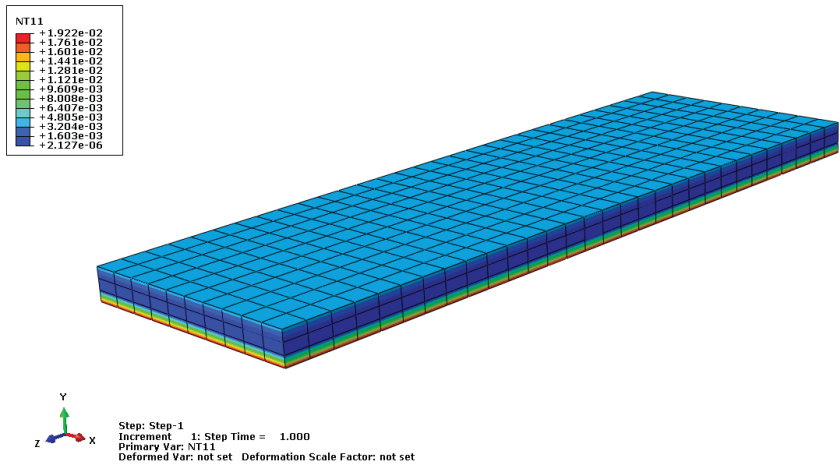
Fig. 7.40 Temperature contour of the slab after analysis.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

7.9.3   Analysis

One thing that needs to be mentioned here is that for the 3D solid model, the absolute zero temperature and the Stefan–Boltzmann constant (please refer to Equation 7.3) need to be defined before an analysis can be performed.

- In the *Model* tree, right click *Model – Edit Attributes*, a window will pop up as shown in Figure 7.39; set the value for both the absolute zero temperature and the Stefan–Boltzmann constant.

For the 2D model, the above parameters are not required.

7.9.4   Model results

Model results in the form of temperature distributions throughout the slab are shown in Figure 7.40.

The temperature output from the slab at the top and bottom surface and at the quarter points can be extracted using *XYData* under the model tree as shown in Figure 7.41.

Then choose *ODB History* output (Figure 7.42) and the temperature against time curve at each selected point on the slab can be plotted.

7.9.5   Other type of slabs

For a typical metal deck composite slab used in steel frame buildings, heat transfer analysis can also be achieved. To save on computation time, rather than a 3D model a 2D model can be built; the modelling results are shown as follows in Figures 7.43 and 7.44.
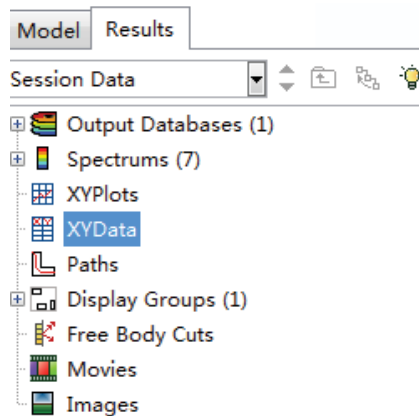
Fig. 7.41   Creating XY data.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
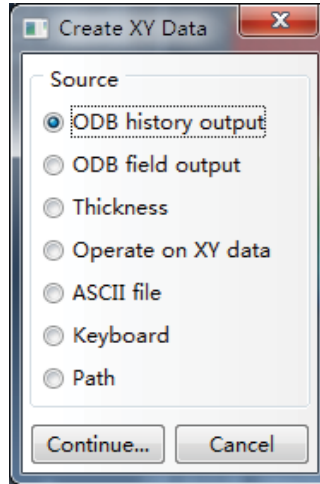
Fig. 7.42   Choosing ODB history output.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Fig. 7.43   Temperature contour of the metal deck composite slab.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
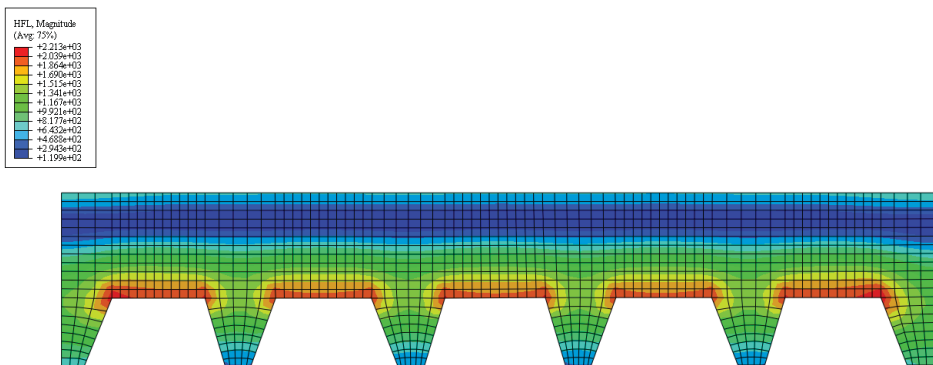
Fig. 7.44   HFL contour of the metal deck composite slab.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

References

Custer, R., Meacham, B.J. 1997. *Introduction to Performance-based Fire Safety*, Society of Fire Protection Engineers.

Building codes and regulations

BS5950, British Standards Institution. 1990. The structural use of steelwork in buildings, Part 8: code of practice for fire resistant design.

BS 476-20, British Standards Institution. 2011. Fire Resistance Test to Building Material,

BSI – 'BS7974: 2002. Code of Practice on Application of Fire Safety Engineering Principles to the Design of Buildings', The British Standards Institution, 2002.

BSI – 'PD7974-3: 2003. Application of fire safety engineering principles to the design of buildings — Part 3: Structural response and fire spread beyond the enclosure of origin (Sub-system 3)', The British Standards Institute, 2003.

BS EN 1991-1-2: Eurocode 1: Actions on Structures-Part 1-2: General actions – Actions on structures exposed to fire., British Standards Institution, 2002.

BS EN1992-1-2 2004. Eurocode 2, Design of concrete structures—Part 1-2: General rules—Structural fire design. Commission of the European Communities

BS EN 1993-1-2 2005. Eurocode 3. Design of steel structures, Part 1-2; general rules. Structural fire design. Commission of the European Communities.

BS EN 1994-1-2 2005. Eurocode 4. Design of composite steel and concrete structures, Part 1-2: general rules. Structural fire design. Commission of the European Communities.

ISO 834-1 1999. Fire-resistance tests – Elements of building construction – Part 1: General requirements

UBC. 1997. Uniform Building Code, International Conference of Building Officials, Whittier, California, USA.

8 Space structures

8.1 Introduction

Due to their visual beauty, advantages in providing long span areas without any obstruction and their being relatively economical, simple and fast to construct, space structures have been widely used in different types of structures consisting of long span to mid-span frames, but also short enclosures, closed roofs, floors, exterior walls and canopies. They can be used in a variety of building types, such as: sports arenas, exhibition pavilions, assembly halls, transportation terminals and stations, airplane hangars and warehouses. Some famous engineering examples include the Birds Nest stadium in Beijing (as shown in Figure 8.1) and Wembley Arena stadium in London. Therefore, knowledge of how to design and analyse space structures is essential for all structural engineers.

Space structures consist of a large number of structural members and their geometry is more complicated compared to conventional structures, therefore they are very difficult to analyse manually. Thus, modelling using computer programs is an effective way in design practice. In this chapter, different types of space structures, as well as basic analysis theory such as buckling analysis and geometric nonlinear analysis, will be introduced. Modelling examples will be given to demonstrate how to model this type of structure in advanced programs such as SAP2000.

8.2 Type of space structures

In design practice, there are several types of space structures that are widely used. In this section, the features of these different types of structures will be introduced.

8.2.1 Double layer grids

Double layer grids are one of the most popular space structures. They have a flat surface and consist of a top and bottom layer of planar networks with the use of inclined web members. These types of space frames are normally designed to be only able to resist tension or compression, as the joints are detailed in such a way that they have no moment or torsional resistance. Therefore, in analysis, the two ends of each member are normally modelled as pinned by defining the moment release or using the TRUSS

Fig. 8.1   Birds Nest stadium designed for the Beijing 2008 Olympic Games.
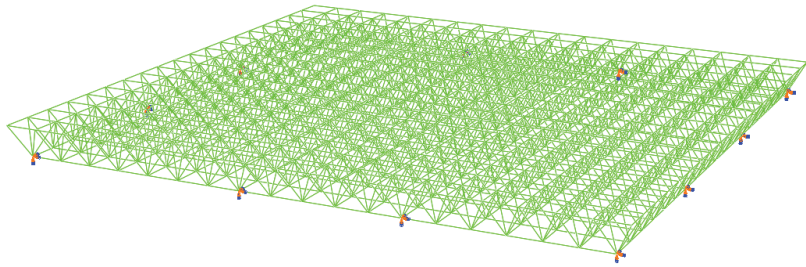
Fig. 8.2   Double layer grids model in Abaqus®.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

elements in most computer programs such as SAP2000 or Abaqus®. Figure 8.2 is a modelling example of a double layer grid space structure using Abaqus®.

The basic elements of a double layer grid are: a planar lattice truss, a square-based pyramid that is part of the octahedron, and a triangular-based pyramid that is part of the tetrahedron (Lan, 2005). Different configurations of top and bottom layers and changing the possible arrangement – such as the dimension of the top layer grid proportional to the bottom layer grid – can make different grid types.

In design practice, as the grid type will affect the overall cost and speed of construction, an engineer should determine it in the early stages of design. The costs involved are the usage of the steel and the manufacturing cost of the joints. Therefore, reducing the joint numbers and joint types as well as member sizes is an effective way of saving cost for this type of structure.

8.2.2   Latticed shell structures

Latticed shell is also one of the most popular space structures. One example can be seen in Figure 8.3. It can be built either as a single layer or double layer grid. For a long span single layer grid, the connections are normally designed to provide rigidity.

Fig. 8.3   Latticed shell structures: Incheon airport Korea.

Therefore, the joints are detailed in such a way that they can resist internal moment and torsional as well as axial loading. However, for a double layer grid, due to its greater redundancy compared to the single layer shell, the joints can be designed as pinned.

Due to the inherent curvature, latticed shells provide greater stiffness than other types of space structure. As they can be built as a single layer, this also reduces the costs and building complexity. Single layer latticed grids are usually composed of smaller span members, as larger spanned members are more likely to buckle due to compressive axial force.

Due to the requirement of clients and architects, nowadays the shell structures are designed with a more complete geometry shape, such as hyperbolic parabaloid shells, or even some very unique shapes, and it is hard, therefore, to classify them. Here, some traditional lattice shell types will be introduced.

Domes

Domes are one commonly used type. Due to their ability to satisfy a maximum amount of space with minimal surface area, they become a very economical option. They are generally composed of compressive forces in the major parts and tensile stresses near the edge, as well as small bending moments at the junction of the shell. They can exhibit many curve types, such as parabolic, spherical and elliptical, covering both polygonal and circular areas.

The most commonly used braced domes are ribbed domes, schwedler domes, three-way grid domes, lamella domes and geodesic domes. The other type of shell that is a widely used for many buildings, especially stadia, is the spherical dome. It is composed of a spheroid parabolic shaped surface, and its common form is the synclastic surface. Figure 8.4 and Figure 8.5 illustrate examples of the two domes with different configurations.
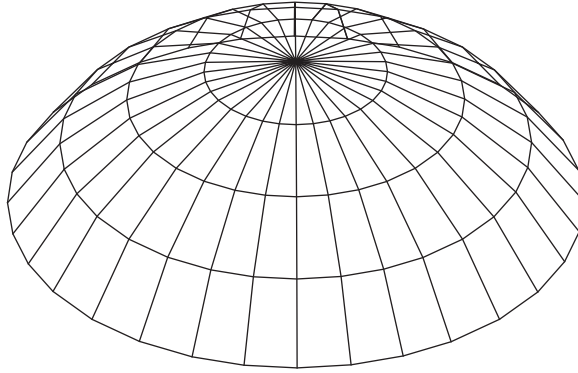
Fig. 8.4   Ribbed dome.
Drawn in AutoCAD.
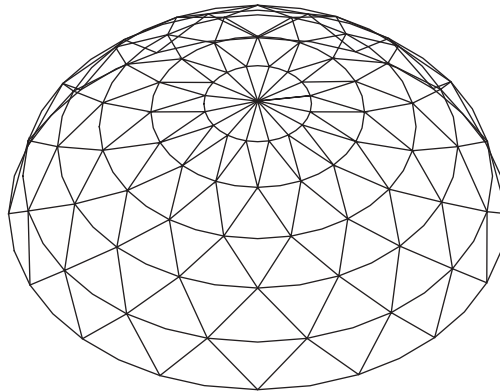
Fig. 8.5   Lamella dome.
Drawn in AutoCAD.

Barrel vault

The so-called barrel vault is a latticed shell structure widely used in stations or shopping malls. It features forms of a cylindrical shell with zero curvature, making its surface easily modifiable. The location and type of supports between the members influences the vault's structural behaviour.

Figure 8.6 and Figure 8.7 give examples of two barrel vaults with different configurations. The major different between them is that the first model does not have tie members between the supports at two sides, while the other one has. Tie members are sometimes employed to help the vaults resist horizontal thrust force.

8.2.3   Tensegrity domes

In the process of designing long span space structures, how to reduce the self-weight of the structure and consequently the cost of the building is a key issue. Among different types of structures introduced in the above sections, the 'Tensegrity System' – a
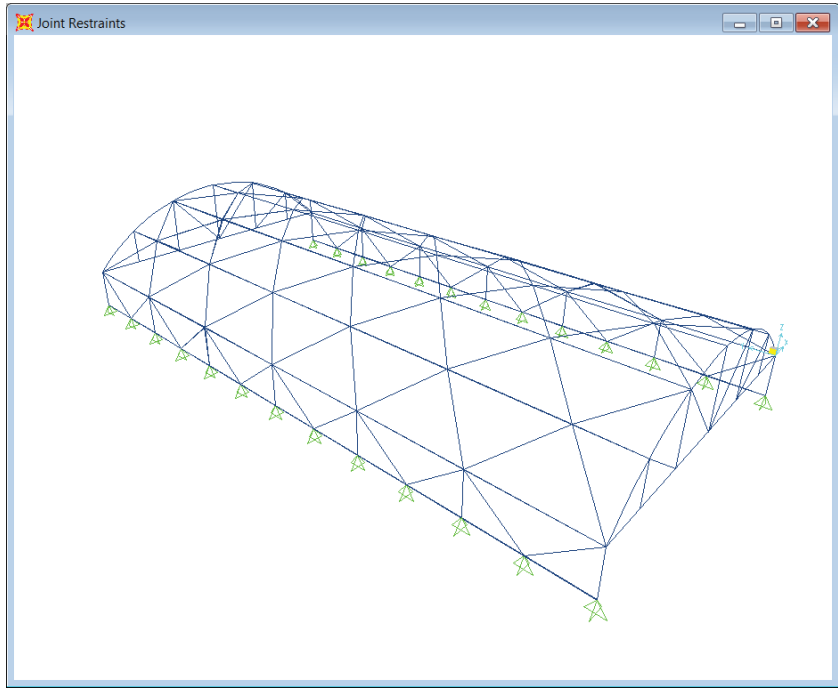
Fig. 8.6   SAP2000 model of a barrel vault.
SAP2000 screenshot reprinted with permission of CSI.
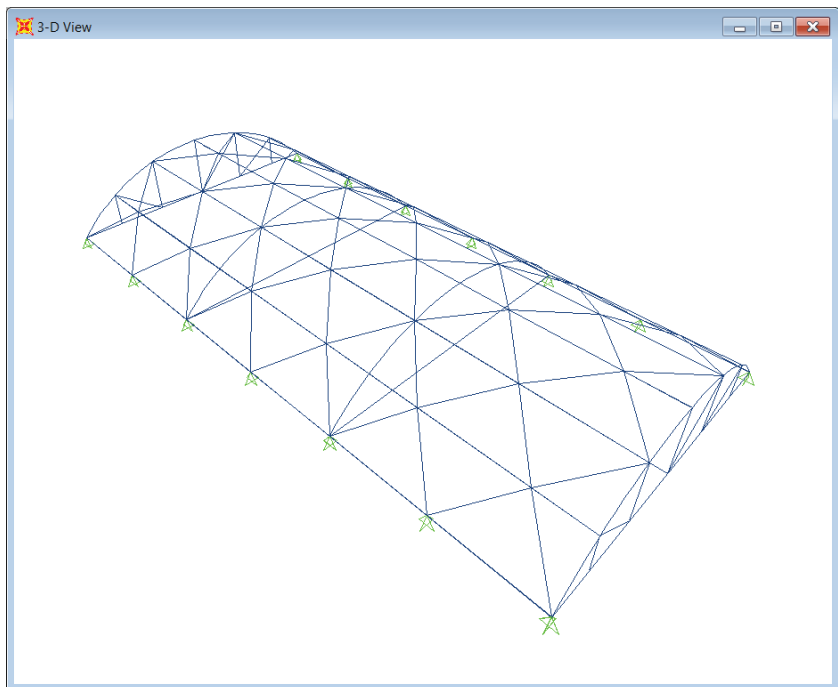
Fig. 8.7   SAP2000 model barrel vault.
SAP2000 screenshot reprinted with permission of CSI.

Fig. 8.8   Georgia dome.
Photo from Structurae: International Database for Structural Engineering, courtesy of Wilhelm Ernst & Sohn Verlag.

self-equilibrium system composed of continuous prestressed cables and individual compression bars – is one of the most promising solutions.

The concept of 'Tensegrity' was first conceived by B. Fuller (1975) and reflected his idea that 'nature relies on continuous tension to embrace islanded compression elements'. D. H. Geiger et al. (1986) made use of Fuller's thought and designed an innovative 'cable dome' in the circular roof structures of the Gymnastic and Fencing Arenas for the Seoul Olympic Games in 1986. However, these cannot be considered an actual Tensegrity system, because the compressed ring is not inside the set of cables. In 1992, Levy (1990, 1991) further improved the layout of the cable dome and built the Georgia Dome in quasi-elliptical shape for the Atlanta Olympic Games. Motro (1992) and Hananor (1993) did research on a double layer grid Tensegrity system. Figure 8.8 shows a photo of Georgia dome, which was the first Tensegrity dome to be built in the world.

8.3   Design load

In this section, the design loads that need to be considered for space structures will be introduced in detail.

8.3.1   Dead load

For a space structure, the dead loads that need to be considered are: the weight of the space frame and cladding, supported lighting, heat and ventilation equipment and so on.

8.3.2 Live load

The normal live loads applied on a space structure are snow, rain, drift load and wind. Among them, wind loads usually represent a significant proportion of the overall forces acting on the frame. Detailed provisions for determining wind loads are given in codes such as SEI/ASCE 7-10 (2010) and Eurocode 1 (2004). They all provide the pressure coefficients and pressure distributions for 3D curved space frames. In design practice, for important long span space structures or those peculiar shapes, it is imperative to undertake further wind tunnel tests in an attempt to establish more accurately the pressure distribution over the roof surface.

8.3.3 Temperature effect

Thermal expansion and contraction due to changes in temperature is another important factor needs to be considered during the design of the space structure. Additional axial loads will also be developed in the structural members if they are restrained. Therefore, potential temperature effect must be considered in the design especially when the span is comparatively large. Special care should also be taken in the choice of support locations to minimise the temperature effect.

8.4 Stability analysis of space structures

Compared with other type of structures, such as tall buildings and bridges, stability is one of the most important design issues that needs to be considered in space structure design. For certain types of space structure, such as the lattice shell, stability is of pressing importance.

The following major types of buckling need to be checked during the design:

1. member buckling
2. local buckling of certain members in certain areas
3. global buckling.

8.4.1 Member buckling analysis

Member buckling refers to the individual member becoming unstable. This can be prevented as long as the structural members are designed according to design codes such as the Eurocode or ASCE code. The corresponding design formulas are given in this guidance. The theories behind these design formulas are Euler's buckling theory and the Perry–Robertson equations.

The Euler buckling formula is based on:

1. Perfectly straight column or strut (no crookedness).
2. Load is concentric (no eccentricity).
3. Column or strut is perfectly pinned at both ends.

The buckling load for a member with pin support at both ends is given as follows:

$$P_{cr} = \frac{\pi^2 EI}{L^2} \tag{8.1}$$

Where:
E is the Young's modulus
I is the moment of inertia of the member
L is the member effective length

The critical stress can be worked out as follows:

$$\sigma_{cr} = \frac{\pi^2 E}{(L/r)^2} = \ critical\ stress \tag{8.2}$$

Where:
r is radius of gyration

Euler's buckling formula only applies to the perfect condition of the structural members. In reality, struts or columns under compression will be eccentrically loaded and also have initial curvatures. The buckling stress is inversely proportional to $(Le/r)^2$ as shown in Equation 8.12. However, for Le/r less than some particular value, depending on the material, a column will fail through material failure rather than by buckling.

Therefore, considering the limitation of Euler's theory, the Perry–Robertson equation is more practical and accurate and it gives more realistic buckling stress for columns and struts. It is based on tests done by Perry and Robertson (BS 449-2). It is the basis for the buckling formulation adopted in Eurocode 3 (2004). Using the Perry–Robertson equations, the buckling stress of a member is given by:

$$\sigma = \frac{1}{2}\left[\sigma_y + \left(1 + 0.003\frac{L}{r}\right)\sigma_{CR}\right] - \sqrt{\frac{1}{4}\left[\sigma_y + \left(1 + 0.003\frac{L}{r}\right)\sigma_{CR}\right]^2 - \sigma_y\sigma_{CR}}$$

$$\tag{8.3}$$

Where:
σ_y is the yield stress
σ_{CR} is Euler's critical stress
L is the effective length of the member
r is the radius of gyration

The member buckling design can be checked automatically in programs such as SAP2000 and ETABS.

8.4.2   Local buckling analysis

The local buckling of a space frame is prone to happen in single layer structures such as single layer lattice shells. It consists of snap-through buckling, which takes place at joints. Snap-through buckling is characterised by a strong geometrical nonlinearity; if the load is increased infinitesimally beyond the critical load, the structure undergoes a large deformation into a different stable configuration that is not adjacent to the original configuration – see Figure 8.9.
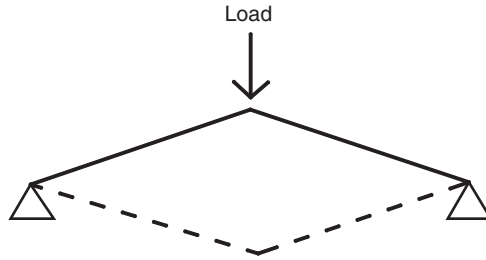
Fig. 8.9 Snap-through buckling example.
AutoCAD.

It is apt to occur when t/R is small,
where:
t is the thickness of the structure and
R is the radius of curvature

Therefore, for a double layer grid structure, this local buckling is less likely to occur due to its larger t/R compared to a single layer structure.

8.4.3 Global buckling analysis

Global buckling refers to a relatively large area of the space frame becoming unstable. It is often triggered by local buckling. The type of buckling collapse of a space frame is greatly influenced by the curvature and the thickness of the structure and the manner of supporting and loading. Therefore, global buckling analysis of the whole structure should also be performed in the analysis.

From existing projects, it has been found that a shallow shell such as a dome is more prone to global buckling than a cylindrical shell. Moreover, single layer space frames exhibit greater sensitivity to global buckling than double-layer structures. For double layer grids, it is normally sufficient to examine the member buckling that may occur in the compressive chord members.

Global buckling analysis searches for the instability modes of a structure with consideration of the P-delta effect under a specified set of loads, such as self-weight, wind, snow and so on. As is widely known, global buckling analysis involves the solution of the generalised eigenvalue problem, which can be performed in most analysis programs such as SAP2000 and ETABS. Here, the basic theories that underpin global buckling analysis are explained as follows:

In the overall structural stiffness equation

$$\{F\} = ([K]_l - [K]_g)\{\Delta\} \tag{8.4}$$

Where:
$[K]_l$ is the normal linear stiffness matrix (without considering the effect of axial force of the member)

$[K]_g$ is the geometric stiffness, it depends on structural geometry (inclusion of P-delta in the SAP2000 and ETABS) and member axial force such as the compression due to the load vector (but also depends on the choice of a particular load case)

If we introduce the so-called buckling factor, λ, this is the scale factor that must multiply the loads **to** cause buckling in the given mode.

$$\lambda\{r\} = \{F\} \qquad (8.5)$$

Where:
r is the load vector

Substituting Equation (8.5) into (8.4):

$$\lambda\{r\} = ([K]_l - \lambda[K]_g)\{\Delta\} \qquad (8.6)$$

Remember, here only $[K]_g$ is affected, as it is related to the load vector. So at buckling,

$$0 = ([K]_l - \lambda[K]_g)\{\Delta\} \qquad (8.7)$$

It can be seen that λ is the diagonal matrix of eigenvalues. The eigenvalue λ is the buckling factor mentioned earlier. It is the scale factor that must multiply the loads in r to cause buckling in the given mode. Therefore, it can also be viewed as a safety factor: if the buckling factor is greater than 1, the given loads must be increased to cause buckling. If it is less than 1, the loads must be decreased to prevent buckling. The buckling factor can also be negative. This indicates that buckling will occur if the loads are reversed.

The buckling analysis of a space structure can be performed using SAP2000 or other programs such as Staad.Pro. In SAP2000, each eigenvalue–eigenvector pair is called a buckling mode of the structure. The modes are identified by numbers from 1 to n and can be found by the program (SAP2000 2013).

It is important to understand that buckling modes depend upon the load. One must evaluate buckling for each set of loads of concern. When using SAP2000, one can create linear buckling Load Cases. For each case you specify a combination of one or more Load Patterns that make up the load vector r. You may also specify the number of modes to be found and a convergence tolerance. A minimum of six modes is recommended by SAP2000 (2013). This will be introduced in detail in the modelling examples in Section 8.5.

8.5   Modelling example of a single layer dome using SAP2000 (including global buckling analysis

In this section, we are going to demonstrate how to set up a single layer dome model and analyse it using SAP2000; the global buckling analysis will also be covered.
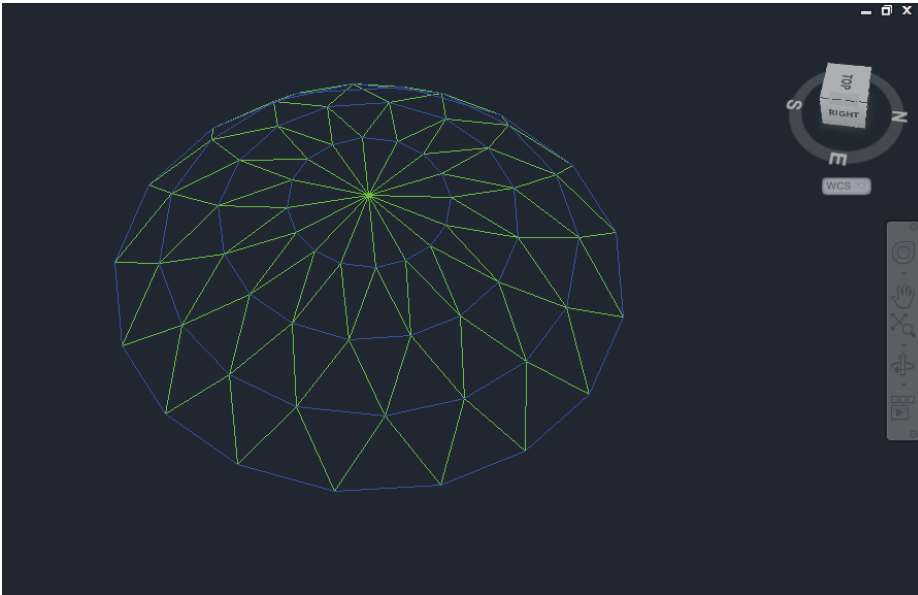
Fig. 8.10   3D AutoCAD model.
AutoCAD.

8.5.1 Set up a 3D model in AutoCAD

In AutoCAD, the geometry of the dome is created as a 3D model as shown in Figure 8.10. In the AutoCAD model, it is important to make sure you have moved the different types of structural members into different layers, as they will be imported into SAP2000 sequentially.

8.5.2 Import the 3D model into SAP2000

Similar to that mentioned in other chapters, after importing the structural wire frame sequentially from different layers of AutoCAD (as shown in Figure 8.11), it is convenient to put members from each layer in the same group. This will facilitate future modification of the member.

8.5.3 Define load pattern

- Go to *Define*, *Load Cases*, and define the load cases as shown Figure 8.12.

8.5.4 Define load cases (including global buckling analysis)

- Go to *Define Load Cases*, and define the load cases as shown Figure 8.13.
- Go to *Define*, *Add New Load Cases*, in *Load Case Type* choose *Buckling*, in the *Load Pattern* add the load cases with the load combination factors.
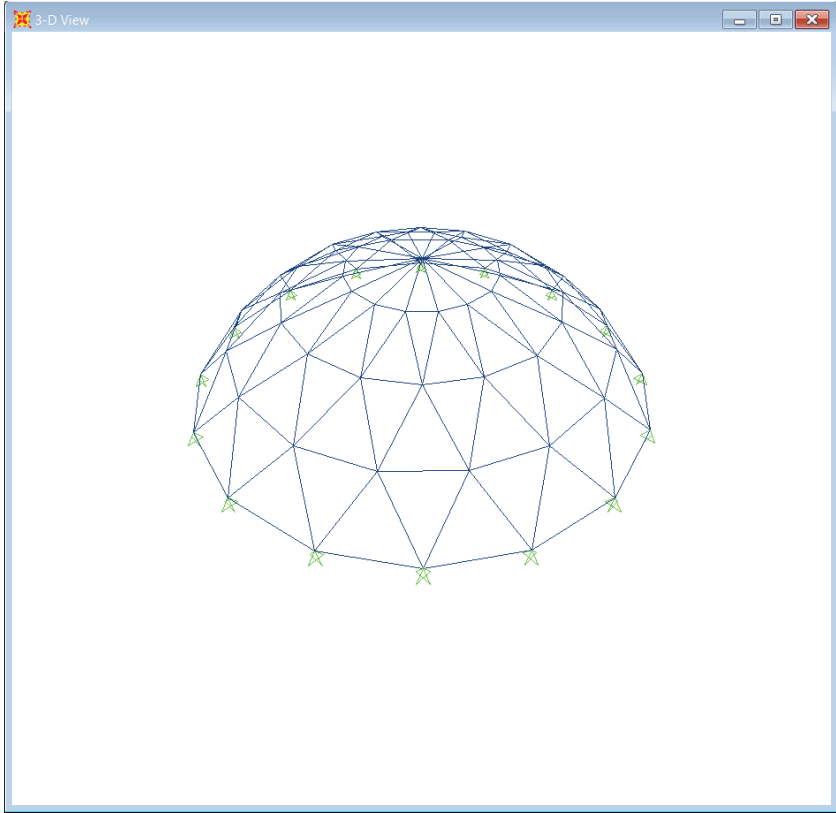
Fig. 8.11   3D SAP2000 model.
SAP2000 screenshot reprinted with permission of CSI.
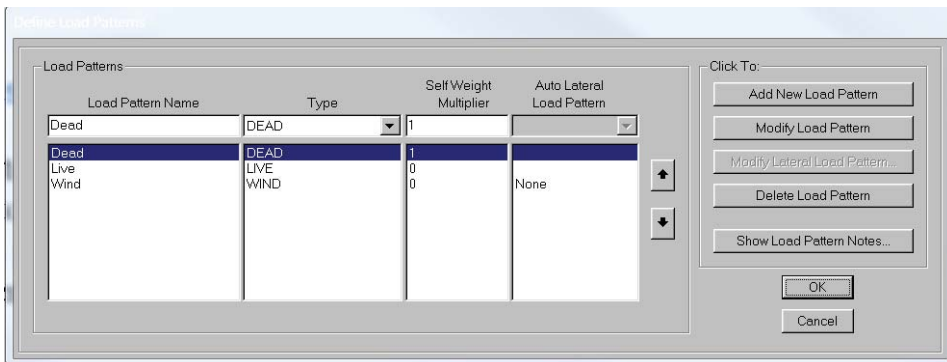
Fig. 8.12   Defining load patterns.
SAP2000 screenshot reprinted with permission of CSI.
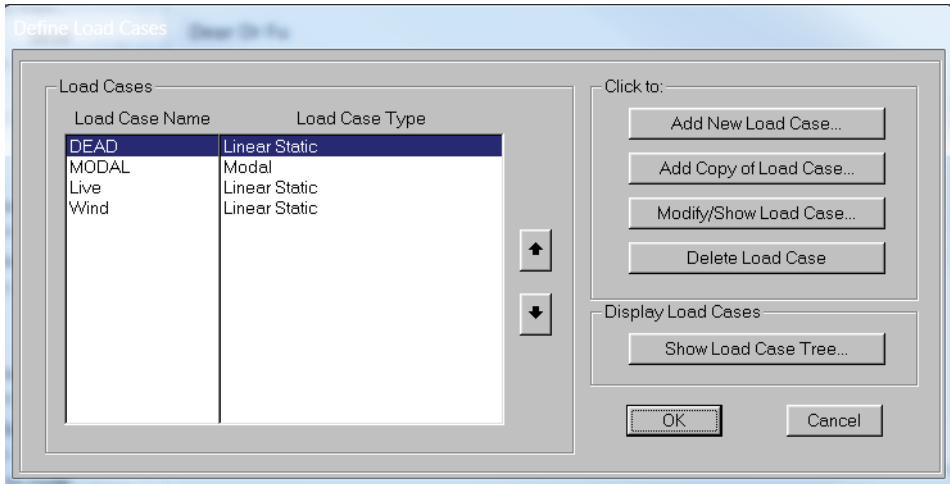
Fig. 8.13   Defining load cases.
SAP2000 screenshot reprinted with permission of CSI.
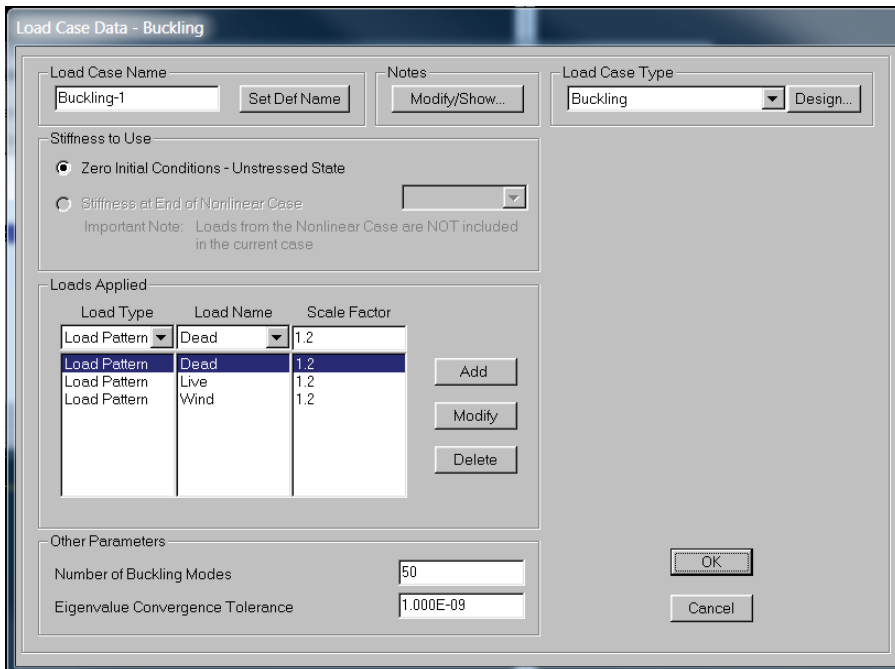
Fig. 8.14   Buckling case 1.
SAP2000 screenshot reprinted with permission of CSI.
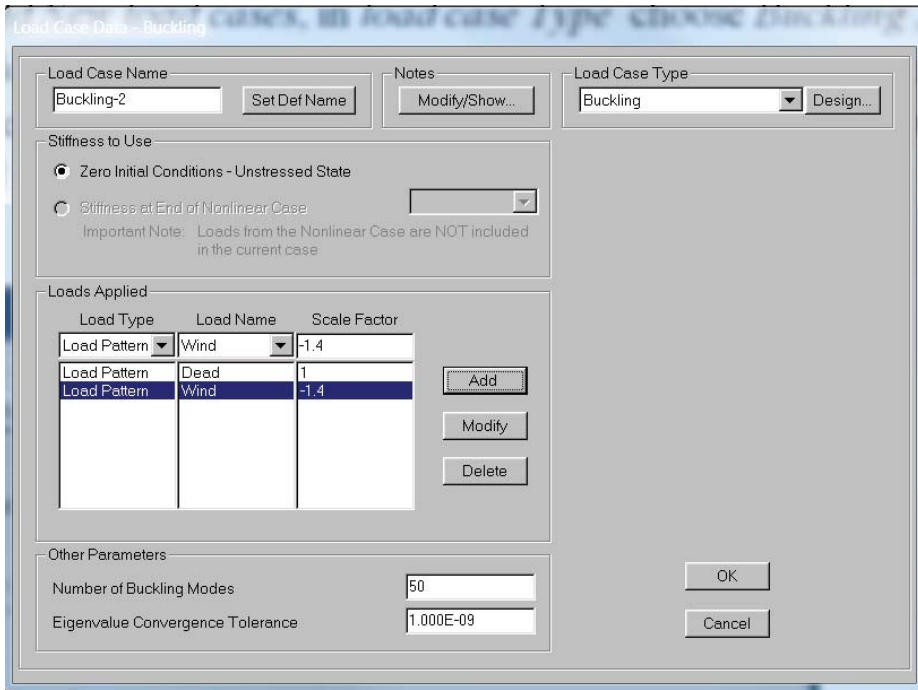
Fig. 8.15   Buckling case 2.
SAP2000 screenshot reprinted with permission of CSI.

- For long span space structures, different load combinations need to be considered as the wind can be either downward or uplift, thus different buckling cases should be defined accordingly as shown in Figure 8.14 and Figure 8.15.
- Here we choose 50 buckling modes to be solved in the *Number of Buckling Modes* option, however at least 6 modes should be selected, as mentioned earlier.
- Click *OK*, then the load cases are defined as shown Figure 8.16.

8.5.5   Run global buckling analysis

- Here, run the two buckling analysis cases we defined. Then click on *Show Deformed Shape*.
- A new window will pop up, as shown in Figure 8.17; choose *Buckling-1* and click *OK*.
- Check the buckling factors.

  As aforementioned, the buckling factor can be viewed as a safety factor:

- If the buckling factor is greater than 1, the given loads must be increased to cause buckling.
- If it is less than 1, the loads must be decreased to prevent buckling. The buckling factor can also be negative. This indicates that buckling will occur if the loads are reversed.
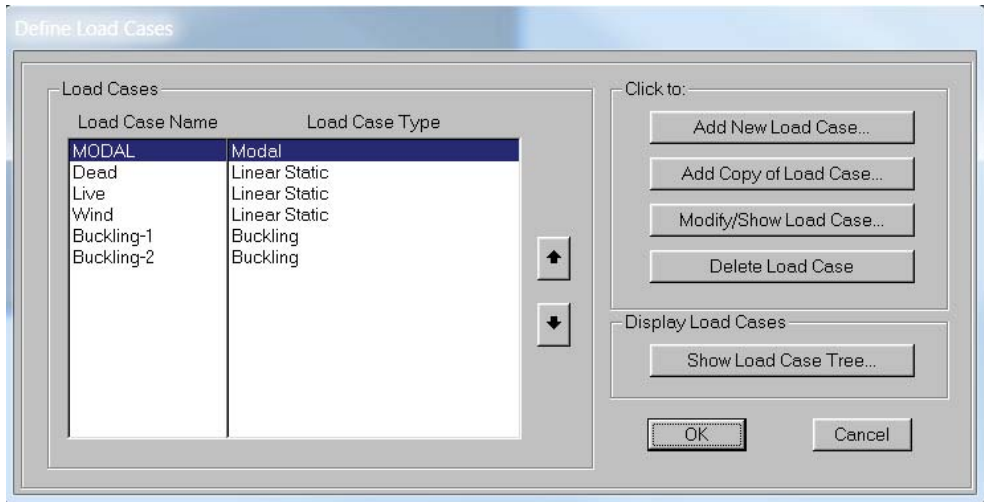
Fig. 8.16   Load cases.
SAP2000 screenshot reprinted with permission of CSI.

Fig. 8.17   Selecting buckling 1.
SAP2000 screenshot reprinted with permission of CSI.
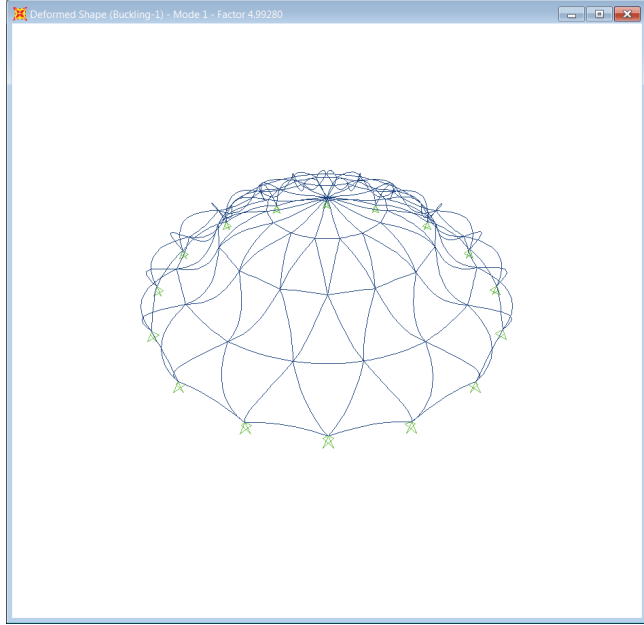
Fig. 8.18   Buckling-1 (buckling-factor = 4.99).
SAP2000 screenshot reprinted with permission of CSI.
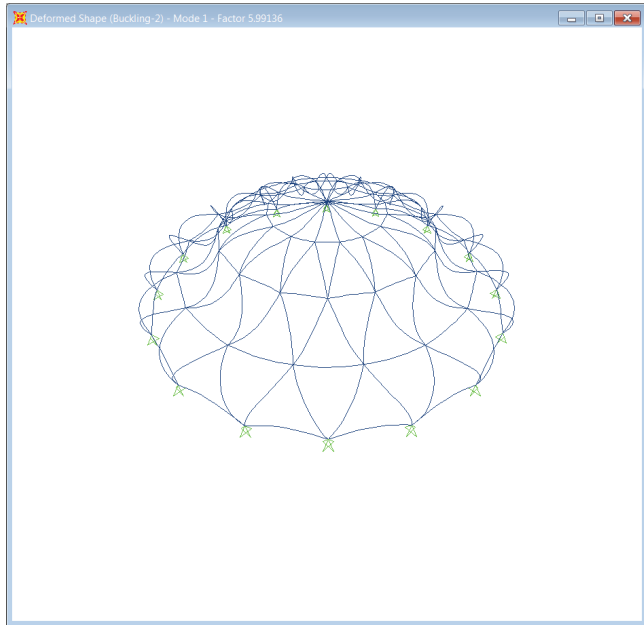
Fig. 8.19   Buckling-2 (buckling-factor = 5.99).
SAP2000 screenshot reprinted with permission of CSI.
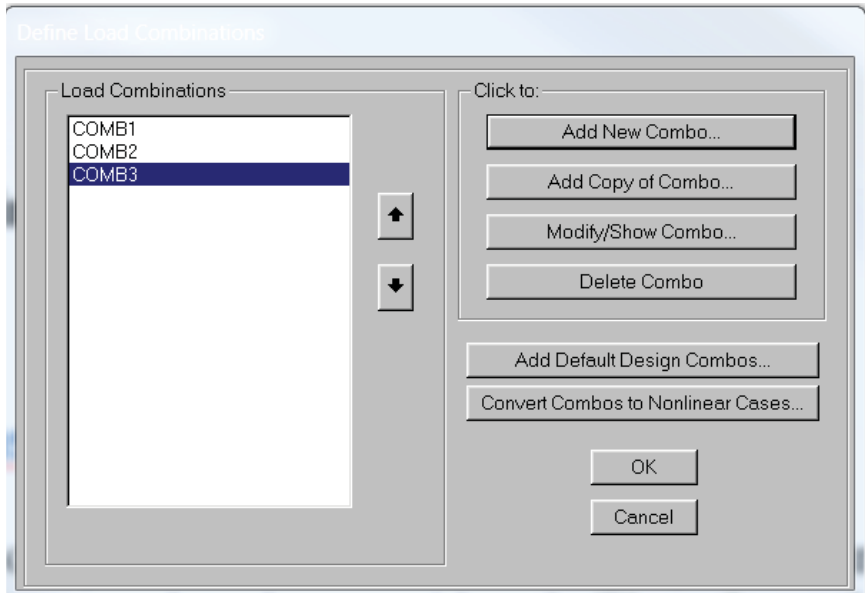
Fig. 8.20   Load combination.
SAP2000 screenshot reprinted with permission of CSI.

Here, as shown in Figure 8.18, for the first load combination buckling-1, the fundamental buckling mode has a buckling factor of 4.99. It is greater than 1, this means that buckling won't occur.

For the second load combination, buckling-2, (Figure 8.19) the fundamental buckling mode has a buckling factor of 5.99. It is greater than 1 so this means that buckling won't occur either. According to the situation of their projects, readers can define different load combinations for buckling analysis.

8.5.6   Define load combination

Apart from global buckling analysis, one also needs to perform a usual static or dynamic analysis to determine the member sizes, therefore the load combination under different load cases needs to be defined.

- Go to *Define*, choose *Load Combination*, and define the required load combination as shown in Figure 8.20. Remember that the direction of the wind needs to be considered.

8.5.7   Analysis and result

- Go to *Show Forces/Stress* and check the force or stress of each structural member as shown below.
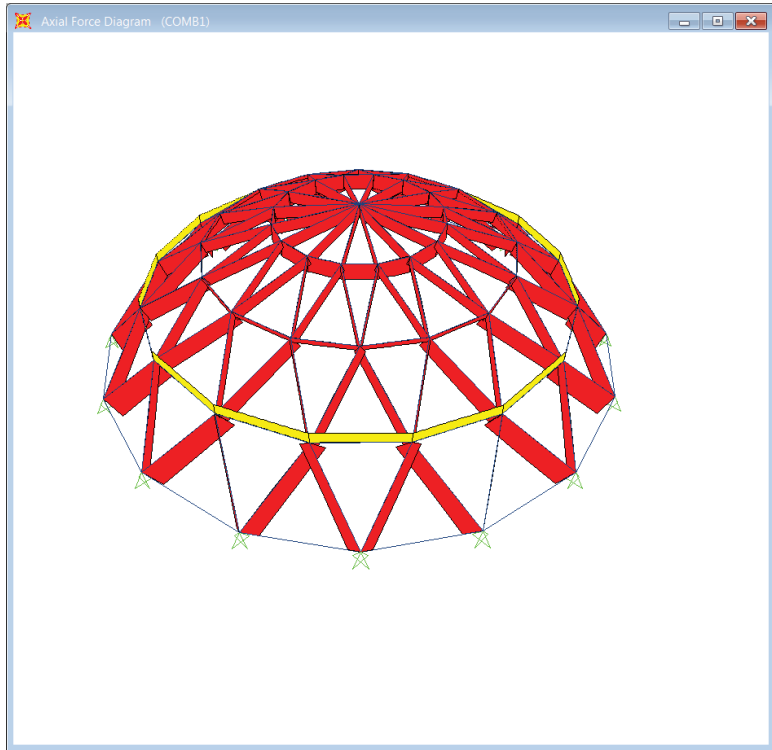
Fig. 8.21   Result of axial force.
SAP2000 screenshot reprinted with permission of CSI.
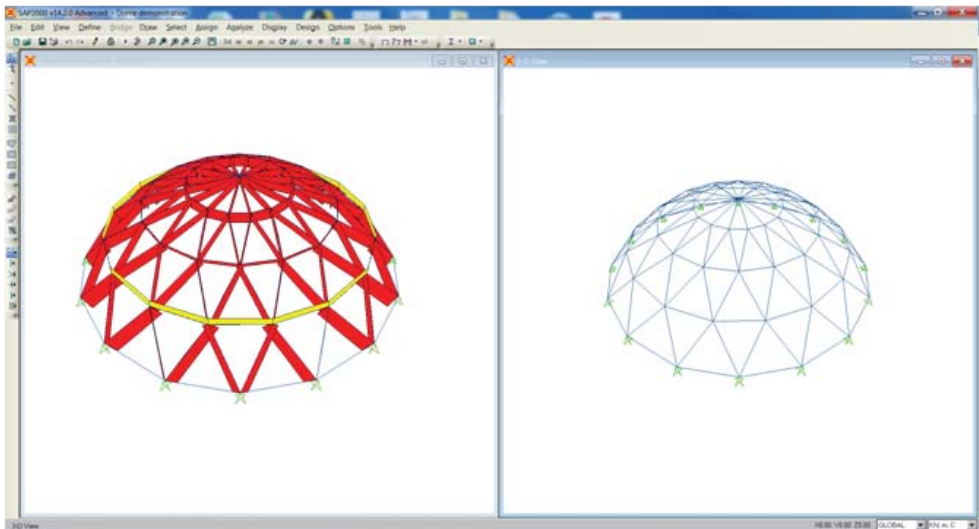
Fig. 8.22   Auto-design of the dome.
SAP2000 screenshot reprinted with permission of CSI.
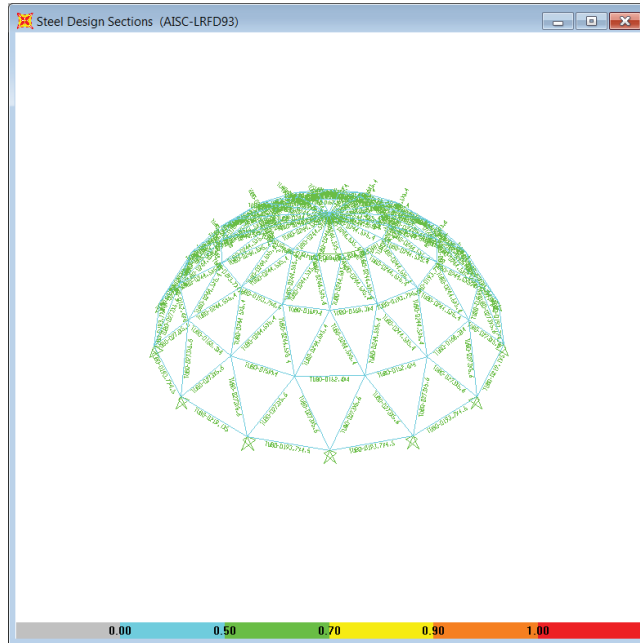
Fig. 8.23 Auto-steel design results.
SAP2000 screenshot reprinted with permission of CSI.

8.5.8   Auto-design module

Engineers can also let SAP2000 design the member size for them.

- Go to *Design*, then *Steel Frame Design*, choose *Start Design/Check of Structure*, as shown in Figure 8.22.
- You can check the auto-design results as shown in Figure 8.23; if the colour of the member is red, this means the member failed.

8.6   Nonlinear geometric analysis of Tensegrity structures

Unlike other types of space structure, the Tensegrity structure is a geometrical non-linear system, and large deformation need to be considered. Therefore, the structural analysis can be divided into two phases (Fu, 2005): the first phase is the initial equilibrium; the second phase is static analysis. The two phases are presented below.

8.6.1   The initial geometrical equilibrium (form finding)

When the boundary condition is determined, the distribution and magnitude of the prestressed force applied to the Tensegrity structure corresponds to the initial geometrical equilibrium of the structure. Therefore, determination of the initial geometrical equilibrium (also called form finding) is important and an iterative method is used, which is introduced here.

During the procedure of determination of the initial equilibrium, the coordinate can be firstly presumed with an ideal distribution of the prestressed force. But the node in the structural grid will not be balanced under this condition; an imbalance force will result and hence the displacement of the node will also result. Thus, the coordinate and the prestressed force need to be adjusted step by step until the whole structure is balanced.

The formula to determine the initial equilibrium is:

$$[K]^0 \{\Delta U\}^0 = -\{P\}^0 + \{R\}^0 \tag{8.8}$$

Where:
$[K]^0$ is the initial stiffness matrix
$\{\Delta U\}^0$ is the variation of the coordinate
$\{P\}^0$ is the prestressed force
$\{R\}^0$ is the residual force

8.6.2   Static analysis

After the determination of the initial equilibrium of the structure, the structure can be analysed under the load. The fundamental equation for static analysis is:

$$[K]\{U\} = -\{P\} + \{R\} \tag{8.9}$$

Where:
$[K]$ is the total stiffness matrix
$\{U\}$ is the displacement vector of the node
$\{P\}$ is the load vector
$\{R\}$ is the residual force

The Newton–Raphson approach is used here for finding the solution of the equation. The total load is divided into small increments and the calculation procedure is divided into corresponding steps; for each increment a new $[K]_i$ is used. The nonlinearity is therefore treated as piece-wise linearity and a constant $[K]_i$ is used in all increments. After each iteration, the 'unbalanced' portion of the external force is estimated and applied in the next increment.

For each step:

$$[K]_i \{\Delta U\}_i = \{\Delta P\}_i \tag{8.10}$$

$$\{U\}_i = \{U\}_{i-1} + \{\Delta U\}_i \tag{8.11}$$

Where:
$[K]_i$ is the stiffness matrix when $n = i$,
$\{\Delta U\}_i$ is the increment of the displacement when $n = i$
$\{\Delta P\}_i$ is the imbalance load when $n = i$
$\{U\}_i$ is the displacement when $n = i$
$\{U\}_{i-1}$ is displacement when $n = i-1$

As $\{\Delta U\}_i$ is obtained, $\{\Delta F\}_i$, the increment of the internal force when $n = i$ can be therefore obtained. For each step the internal force can be obtained as:

$$\{F\}_i = \{F\}_{i-1} + \{\Delta F\}_i \tag{8.12}$$

Where:
$\{F\}_i$ is the internal force of member $n = i$
$\{F\}_{i-1}$ is the internal force of member $n = i-1$
$\{\Delta F\}_i$ is the increment of member when $n = i$

When all the steps are finished, the increment of the displacement and the increment of the internal force in different steps will be added together, so the final result can be obtained.

$$\{U\} = \sum_{i=1}^{n} \{\Delta U\}_i = \{\Delta U\}_1 + \{\Delta U\}_2 + \cdots + \{\Delta U\}_n \tag{8.13}$$

$$\{F\} = \sum_{i=1}^{n} \{\Delta F\}_i = \{\Delta F\}_1 + \{\Delta F\}_2 + \cdots + \{\Delta F\}_n \tag{8.14}$$

8.7 Modelling example of Tensigrity dome using SAP2000 (nonlinear geometrical analysis

In this section, we are going to demonstrate how to set up a Tensigrity dome model and analyse it using SAP2000. Nonlinear geometrical analysis will also be covered.

8.7.1 Set up a 3D model in Rhino

The geometry of the Tensegrity structure is sophisticated. It is more efficient to set up the 3D model in software such as Rhino (as shown in Figure 8.24). As the Tensegrity structure consists of cables and struts, it more convenient to put different types of structural members into different layers and import them to SAP2000 sequentially, to avoid chaos in SAP2000.

There are different ways to set up the Tensegrity structures. The unit prisms can be set up first, as shown in Figure 8.25, and then joined together. The contiguous strut configuration (vertex to vertex) is one option because all the joints of the prisms are connected to nearby prisms, and thus the dome is more robust, safe and allows the structure to span a greater distance (shown in Figure 8.26).

8.7.2 Import 3D model into SAP2000

Figure 8.27 shows the model after importing it into SAP2000. After importing the structural frame sequentially from different layers of Rhino into SAP2000, it is normally continent to group the members from each layer into the same group. This will facilitate future modification of the member.
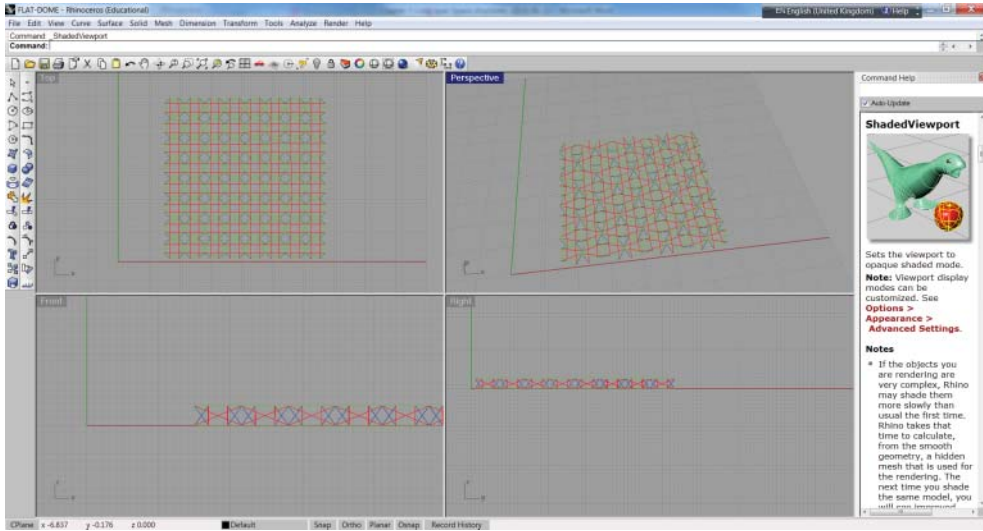
Fig. 8.24   A 3D Tensegrity model in Rhino.
Rhino 3D screenshot reprinted with permission of McNeel.
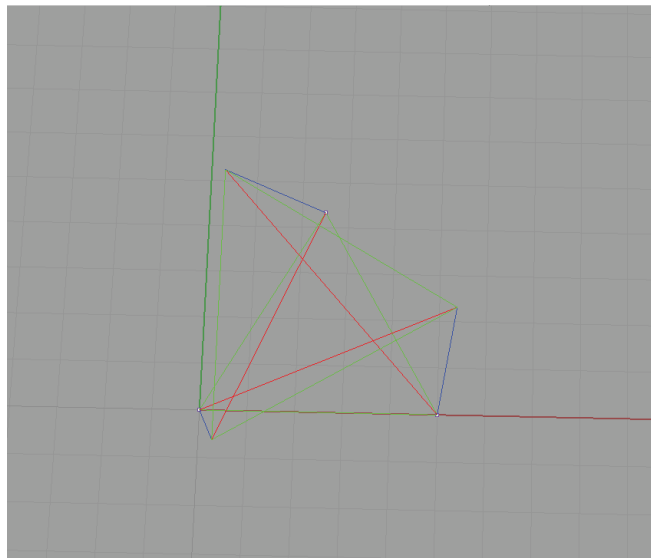
Fig. 8.25   A unit Tensegrity prism in Rhino.
Rhino 3D screenshot reprinted with permission of McNeel.

8.7.3   Nonlinear geometric analysis of Tensegrity using SAP2000

In the analysis of a Tensegrity structure, it is important to consider the geometric nonlinearity; here, we are going to demonstrate how to perform geometric nonlinear analysis using SAP2000.
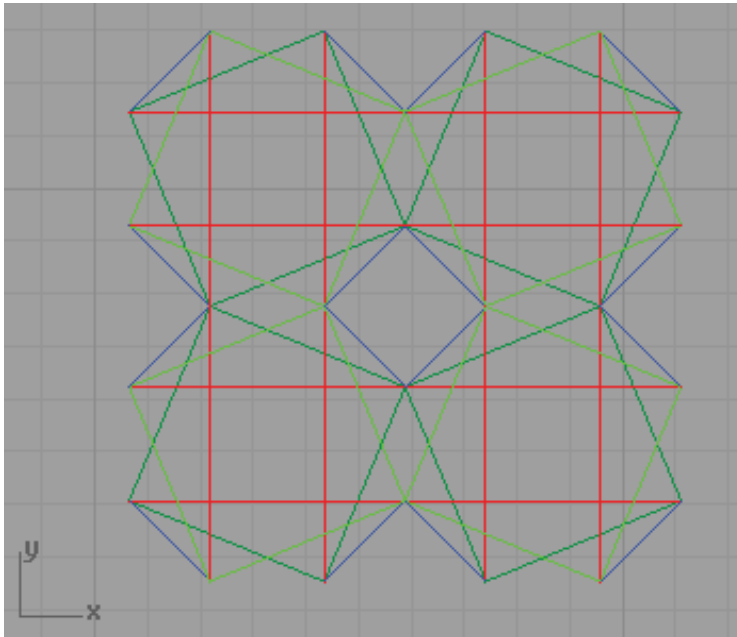
Fig. 8.26   Connecting Tensegrity prisms in Rhino.
Rhino 3D screenshot reprinted with permission of McNeel.
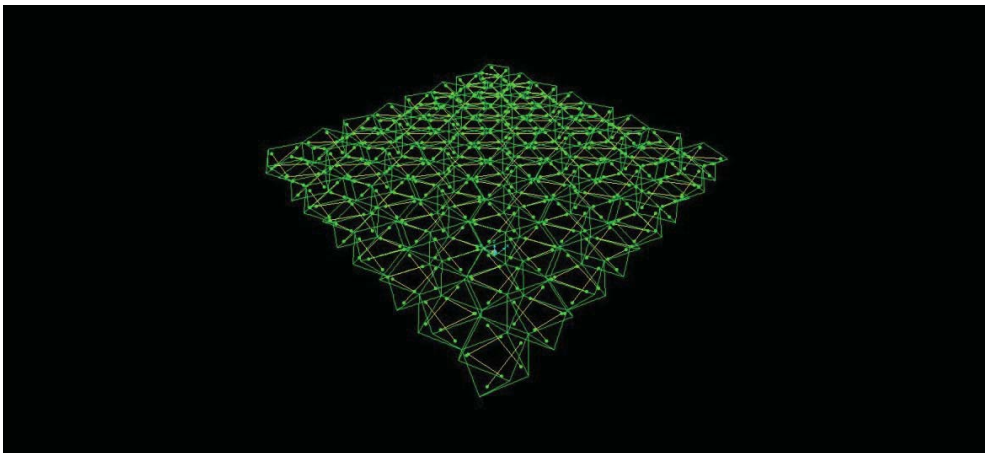
Fig. 8.27   3D Tensegrity model after importing into SAP2000.
SAP2000 screenshot reprinted with permission of CSI.
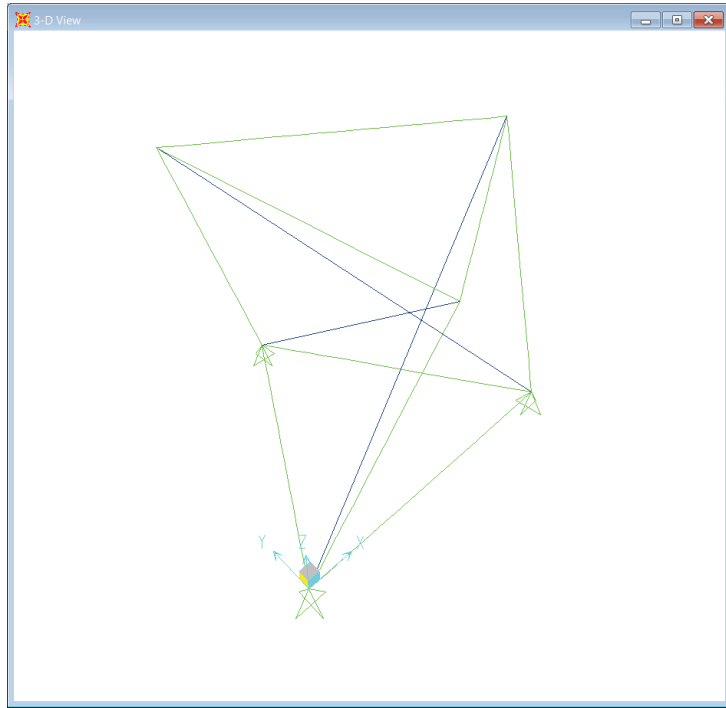
Fig. 8.28 A unit Tensegrity prism in SAP2000.
SAP2000 screenshot reprinted with permission of CSI.

To facilitate the demonstration, we choose a unit prism as shown Figure 8.28; the green lines are defined as cable elements and stand for cables. The yellow lines are defined as frame elements to stand for struts

8.7.4 Define the prestressed force

For a Tensegrity structure, the cables are not slackened as they are prestressed through connecting to the end of the strut. Therefore, in the modelling, it is essential to define the prestressed force at the beginning of the analysis.

There are two ways to define the prestressed force, you can either define the cable geometry or define the target force, and both methods are as follows:

Method 1

Click the *Edit Menu – Edit Lines – Edit Cable Geometry*. A new window will pop up. You can define the cable tension (or prestressed force) here as shown in Figure 8.29.

Method 2

Click the *Define Menu – Load Patterns –* except the *Dead*, *Live* and *Wind Load* patterns, and define a new load pattern called *Target* as it is shown in Figure 8.30.
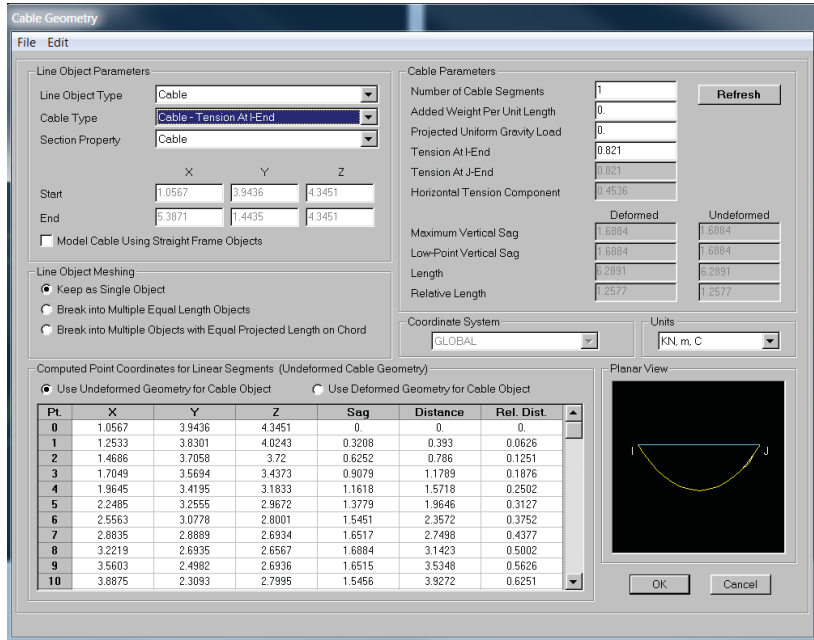
Fig. 8.29   Defining the cable tension.
SAP2000 screenshot reprinted with permission of CSI.
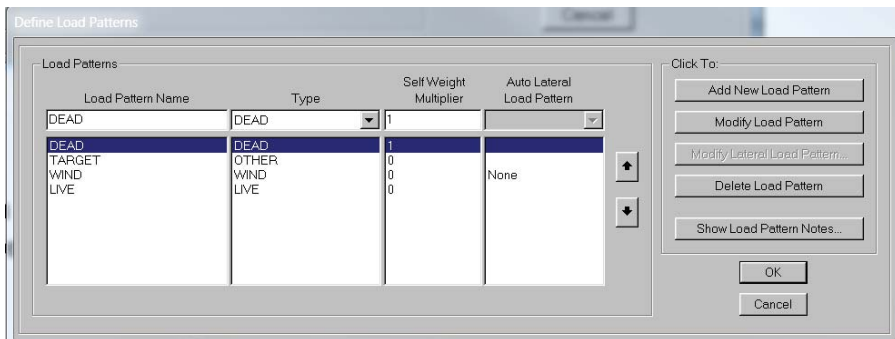
Fig. 8.30   Creating the target load pattern.
SAP2000 screenshot reprinted with permission of CSI.

Select all the cables, go to *Assign Menu – Cable Load – Target Force*, a new window will pop up. Work out the estimated prestress force (for example 80 kN) click *OK*. See Figure 8.31.

8.7.5   Form finding (determination of initial geometrical equilibrium

As mentioned in Section 8.6.1, the first step of the analysis is to determine the initial geometrical equilibrium under the prestressed force, which we call form finding.
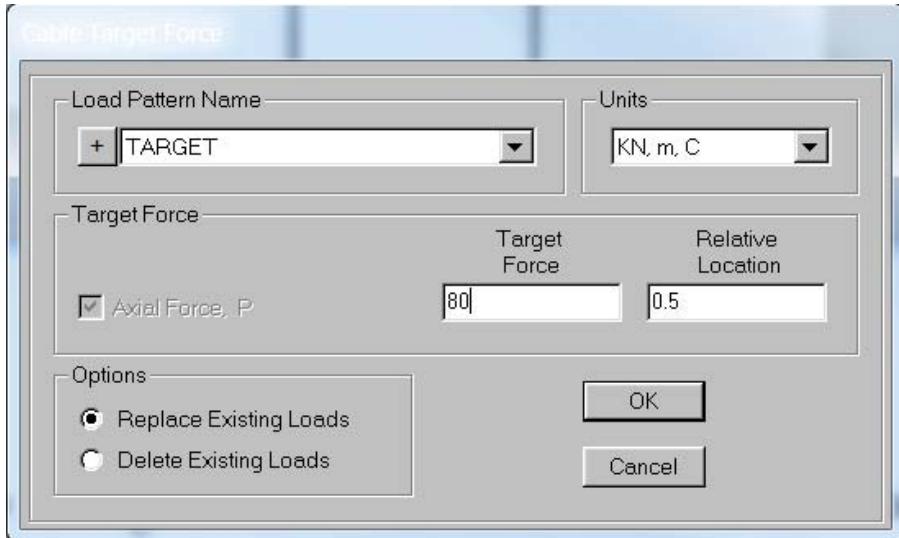
Fig. 8.31 Defining the target force parameter.
SAP2000 screenshot reprinted with permission of CSI.
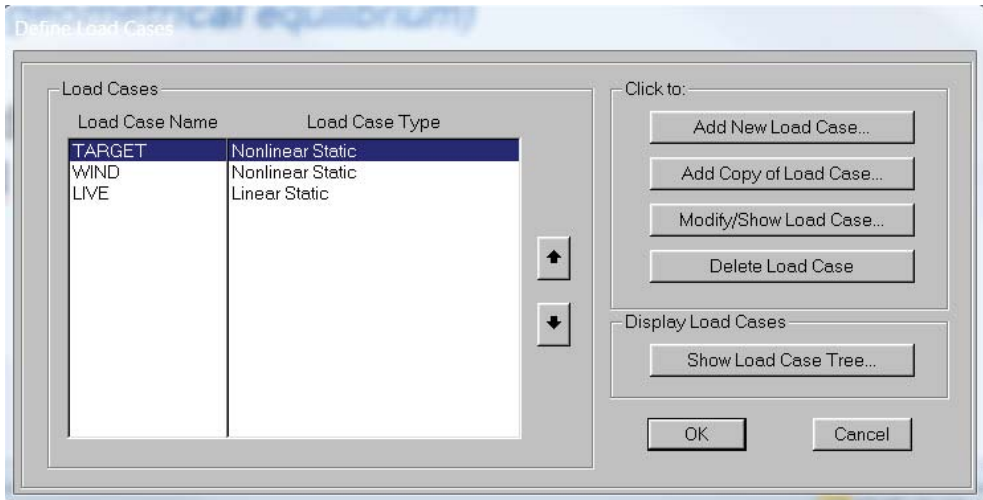
Fig. 8.32 Creating target load case.
SAP2000 screenshot reprinted with permission of CSI.
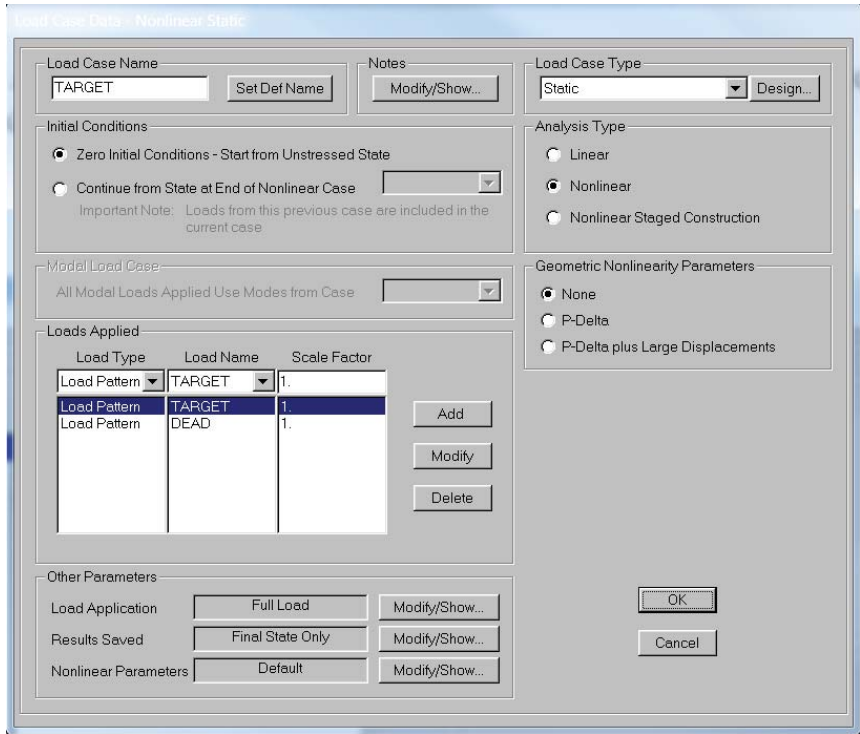
Fig. 8.33   Defining the target load case parameters.
SAP2000 screenshot reprinted with permission of CSI.
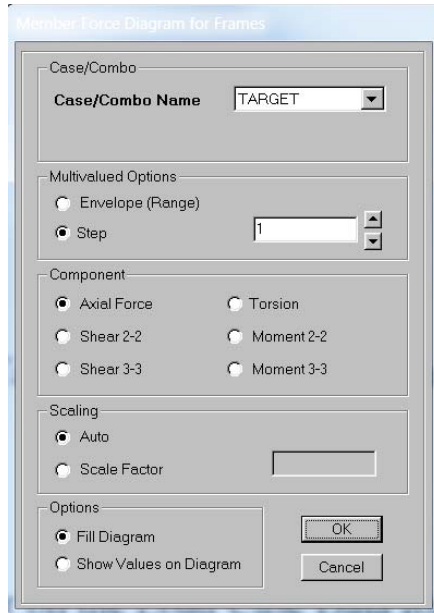
Fig. 8.34   Displaying axial force under the target load case.
SAP2000 screenshot reprinted with permission of CSI.
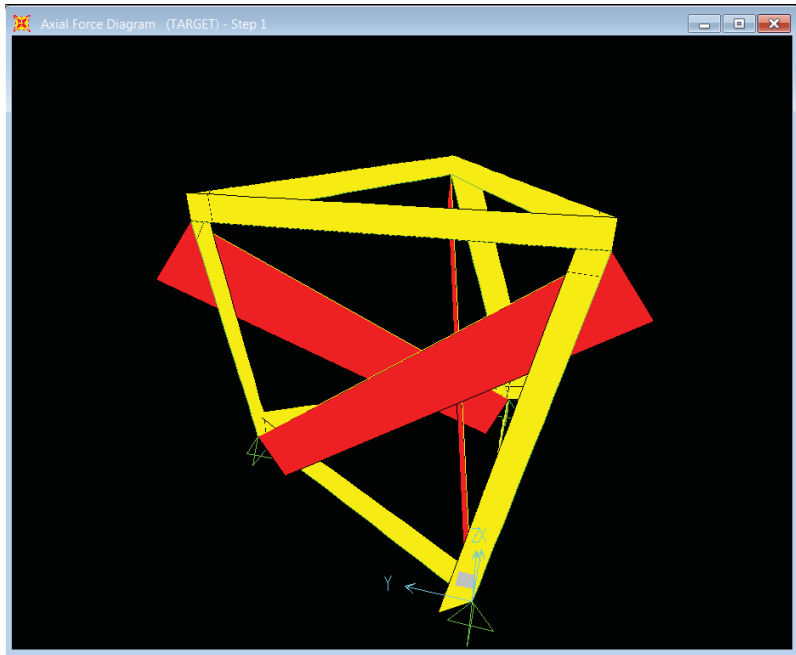
Fig. 8.35   Form finding result.
SAP2000 screenshot reprinted with permission of CSI.
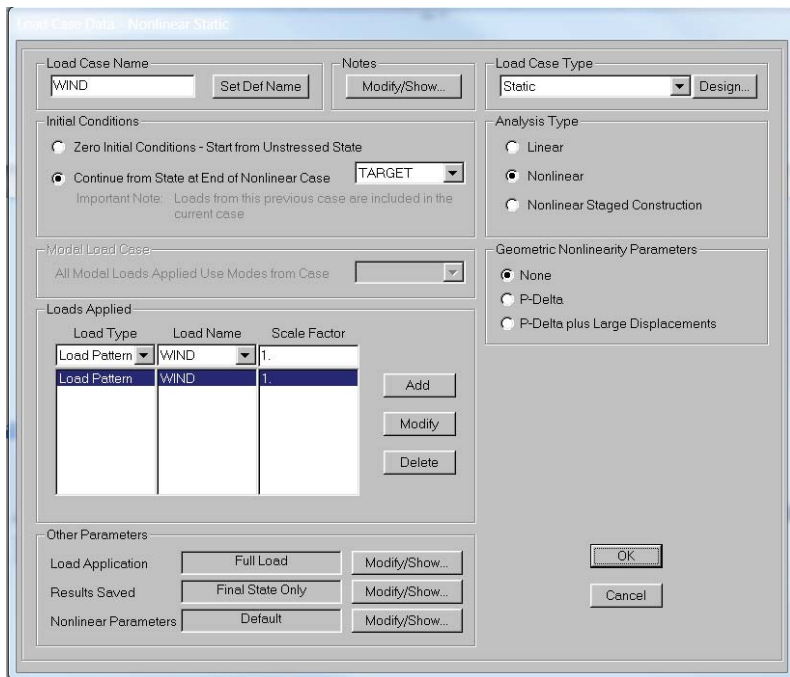
Fig. 8.36   Define the wind load starting from the target load case.
SAP2000 screenshot reprinted with permission of CSI.
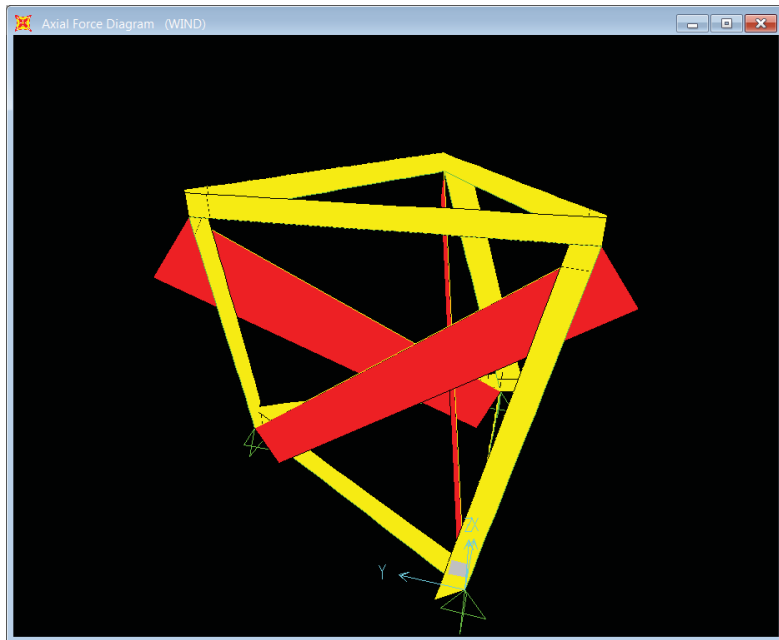
Fig. 8.37   Static analysis result.
SAP2000 screenshot reprinted with permission of CSI.

- Click *Define Menu – Load Cases*.
- Click *Add New Load Case*, and define a new load case *Target*. Set the parameters as shown in Figure 8.33. Choose both the target load and dead load case; this is because the self-weight of the structure should be included in the form finding.
- Run an analysis of the Target load case: click on *Show Forces/Stress* and choose *Target*, select *Axial Force*.
- A new window will pop up as shown in Figure 8.34.
- Check the axial forces of cables under the Target load case to see whether equilibrium has been achieved (see Figure 8.35).

8.7.6 Static analysis

- Click *Add New Load Case*, define a new static load case which is wind. Set the parameters as shown in Figure 8.36; here, it is important to select *Initial Conditions – Continue From State at End of Nonlinear Case – Target* as the static analysis should be based on the result of the form finding.
- Run an analysis of the wind load case: click on *Show Forces/Stress*, choose *Target*, select *Axial Force*, the results are shown in Figure 8.37.

References

Fu, F. 2005. "Structural behavior and design methods of Tensegrity domes," *Journal of Constructional Steel Research*, 61, 1: 23–35.

Fuller R. B. 1975. *Synergetics Explorations in the Geometry of Thinking*. Collier Macmillan Publishers.

Geiger D., Stefaniuk A., Chen D., 1986. "The design and construction of two cable domes for the Korean Olympics, shells, membranes and space frames", Proceedings of IASS Symposium, Osaka, Vol. 2, Elsevier, pp. 265–272.

Hananor, A. 1993. "Developments in tensegrity systems: an overview." In *Proceedings of the 4th Conference on Space Structures*, ed. H. Nooshin, University of Surrey, pp. 987–997.

Lan, T. T. 2005. "Space frame structures," in *Handbook of Structural Engineering*, Second Edition, ed. W.F. Chen, E.M. Lui. CRC Press.

Levy, M. 1991. *Floating Fabric over Georgia Dome*, Civil Engineering ASCE, pp. 34–37.

Levy, M. 1990. "Hypar-Tensegrity Dome", Proceedings of International Symposium on Sports Architecture, Beijing, 1989, pp. 157–162.

Motro, R. 1992. "Tensegrity systems: the state of the art", *International Journal of Space Structures*, 7, 2: 75–81.

Perry Robertson formula (BS 449-2). Retrieved from http://www.roymech.co.uk/Useful_Tables/Mechanics/Struts.html#Perry 22 November 2010.

Building codes and regulations

BS EN 1991-1-4: 2005 Eurocode 1 Actions on structures – Part 1-4: General actions — Wind actions, European Committee for Standardization.

EN 1993 2004 Eurocode 3: Design of steel structures, European Committee for Standardization.

SEI/ASCE 7-10, 2010. Minimum Design Loads for Buildings and Other Structures, ASCE.

Software and manuals

SAP2000 Theory Manual, 2013. Computer and Structures INC.

9 Bridge structures

9.1 Introduction

Bridge structures are widely used in current construction practice. By their usage, they can be classified as rail bridges (one of best known examples is the Forth Bridge in Scotland), foot bridges (one of the most well-known examples is the Millennium Bridge in London) or vehicle bridges (such as the Millau Viaduct in France). They can also be classified based on their structural form. In this chapter, we are going to introduce the different structural types of bridges, and the main issues that need to be considered during their design. We are also going to introduce two modelling examples using SCI Bridge and SAP2000 based on real construction projects: the Milau Viaduct and Forth Bridge. The modelling example of vibration analysis of the Millennium Bridge will be introduced in Chapter 10.

9.2 Structural types of bridges

Based on their load path and structural behaviour, the main structural types of bridges are: beam bridges, truss bridges, cantilever bridges, arch bridges, suspension bridges, and cable-stayed bridges.

9.2.1 Beam bridges and truss bridges

A beam bridge is one of the simplest bridges due to its structural form. The bridge is simply supported at each end of the bridge decks. Thus no moments can be transferred throughout its supports, and in analysis the supports should be modelled as pinned. Common materials used in the beam bridge were stones or wood in ancient times, but in modern infrastructure reinforced concrete or steel are widely used. In steel beam bridges, the main beams can be formed either by trusses or box girders. The box section is widely used for short-span bridges, due to its good resistance to torsional moment.

Truss bridges belong to the beam bridge group but can also be categorised as an independent type of bridge. The truss structure is made from the connection of tension and compression structural members forming triangular units. Figure 9.1 shows an example of a truss beam bridge.

Fig. 9.1   Illustration of a truss beam bridge in London.

9.2.2   Arch bridges

The main structure members form a curved arch. These types of bridges utilise the arch action, which has the ability to transfer the gravity load – such as the dead and live – into a horizontal thrust that is controlled by the abutments of the bridge at each sideway. Therefore, one of the features of this type of bridge is that heavy abutments will be used.

Unlike the beam bridge, which is mainly in bending and shear, the key structure members are mainly in compression due to the arch shape. Traditionally, the materials used for the construction of an arch bridge are stone, brick or concrete, as these types of materials are very strong in compression but weak in tension and bending.

In modern construction, arch bridges made of steel members are also popular. One of the most famous examples is the Sydney Harbour Bridge, as shown in Figure 9.2.

9.2.3   Cantilever bridges

A cantilever bridge span is usually designed with two cantilever arms, which extend from opposite sides and join at the centre of the bridge. One famous project example is the Forth Bridge in Scotland (shown in Figure 9.3) which will be introduced in detail as a modelling example in Section 9.5.

9.2.4   Suspension bridges

The suspension bridge is a type of bridge used for long spans. It can achieve a longer main span than any other type of bridge. One famous example is the Golden Gate

Fig. 9.2   Sydney Harbour Bridge.

Fig. 9.3   Forth Bridge.
Photo from Structurae: International Database for Structural Engineering, courtesy of Wilhelm Ernst & Sohn Verlag.

Fig. 9.4   Golden Gate Bridge in California.
Photo from Structurae: International Database for Structural Engineering, courtesy of Wilhelm Ernst & Sohn Verlag.

Bridge (as shown in Figure 9.4). The main structural components of this type of bridge are: cables, supporting towers and vertical suspender cables that carry the weight of the bridge deck. Sufficient anchorages should be designed to hold the cables of the bridge.

The materials required for construction will be less than any other bridge of the same length and as a result the construction cost will be reduced. It can withstand earthquakes movements better than heavier bridges.

9.2.5   Cable-stayed bridges

Cable-stayed bridges are visually similar to suspension bridges. However, their structural behaviour is completely different. A typical cable-stayed bridge is a continuous girder with towers supported by the bridge's foundation. From these towers, cables extend down diagonally, usually to both sides, in order to support the girder. In the cable-stayed bridge, the towers form the primary load-bearing structure.

Compared to suspension bridges, the main advantage of this type of bridge is that there is no need for anchorages to hold cables. The cables work as a restraint to the bridge deck, providing greater rigidity, thereby reducing its deformation. In construction practice, these types of bridges are ideal for spans that are longer than those of the cantilever bridges but shorter than those of a suspension bridge. For span length up to 1000 m, cable-stayed bridges are considered the most economical.

One famous project example is the Millau Viaduct Bridge in France (shown in Figure 9.5) which will be introduced in detail in as a modelling example in Section 9.4.

Fig. 9.5   Millau Viaduct Bridge.
Photo from Structurae: International Database for Structural Engineering, courtesy of Wilhelm Ernst & Sohn Verlag.

9.3   Structural design of bridge structure

The design procedure of a bridge must take into consideration several important factors in order to reach a best solution. These are choices of bridge systems, materials, dimensions, foundations, aesthetics, local landscape and environment. Structural designers are required to provide the most effective structural solution with maximum safety and minimum cost.

In bridge design, superstructure mainly refers to the deck, beams, trusses and different accessories above the bridge, while substructure mainly refers to the mechanisms of the bridge that support the load from the superstructure and then transfer the load into the ground, such as columns, pier walls and piles.

9.4   Design loading

Design guidance such as Eurocode 1 (2003) or AASHTO (2007) gives detailed requirements of the design load when working on bridges. According to Eurocode 1 (2003), the following should be considered: self-weight and imposed loads, wind, thermal actions, actions during execution, accidental actions (impact loads) and traffic loads. There are also other actions such as concrete creep and shrinkage, settlements and earth pressures, or seismic actions that need to be considered.

Load models are developed for highway bridges in both Eurocode 1 (2003) and AASHTO (2007). The basic load combination for highway bridges is a simultaneous occurrence of dead load, live load and dynamic load. The models are based on the available statistical data on these loads.

In this section the major loads on the bridge superstructures and how these loads are distributed on the bridge will be illustrated. Readers can refer to the design specification mentioned in this chapter for further information.

9.4.1   Dead loads

According to the specifications of Eurocode 1 (2003), the dead loads that can exist in a bridge come from two parts. One is the structural part, such as the structural steel or the structural concrete, and these can be determined based on the dimensions of the structural elements. The other is the non-structural parts, such as concrete barriers, the surface and the deck cover on the bridge and any other elements considered as permanent loads by the engineer.

9.4.2   Live loads

The predominant live loading is traffic load on the bridge. The maximum live loads a bridge can support are based on the maximum number of vehicles that will pass over the bridge at a given time. Future growth of faster and larger moving transport should also be considered. In some cases it is important to use different measures and tests regarding the performance and behaviour of the structure under maximum load.

The effect of live load depends on many parameters including the span length, truck weight, axle loads, axle configuration, position of the vehicle on the bridge (transverse and longitudinal), number of vehicles on the bridge (multiple presence), girder spacing and stiffness of structural members (slab and girders). The effect of these parameters is considered separately (Nowak, 1993).

Eurocode 1 (2003) provides different traffic load models for the designer to use, they are as follows:

- Vertical forces: LM1, LM2, LM3, LM4.
- Horizontal forces: braking and acceleration, centrifugal, transverse.
- Groups of loads: gr1a, gr1b, gr2, gr3, gr4, gr5.

AASHTO (2007) also gives different traffic load model numbers. For example, trucks that have a load configuration larger than the allowed specifications and are named 'Highway Load '93' or HL93.

9.4.3   Seismic effects on bridges

Earthquake load is another important consideration. Ground shaking or deformation is a primary cause of damage to the bridge, resulting in damage or collapse.

Superstructure designs have to be effectually durable to remain principally elastic during earthquakes.

9.4.4 Wind effects on bridges

The well-known collapse of Tacoma Narrows Bridge in 1940 (Billah and Scanlan., 1991) shows the importance of wind effect on long-span bridge performance. Extensive research has been carried out to investigate the effects of wind on long-span bridges such as suspension bridges or cable-stayed bridges and analysis of the response of the bridge under wind load. In certain situations, wind tunnel tests and detailed finite element dynamic models need to be developed.

9.4.5 Accidental actions (impact loads)

In bridge design it is also important to consider accidental actions such as the impact from ships. There are several scenarios of impact cases, such as bow collision with a bridge pillar or side collision or ship deckhouse collision with the bridge span and so on.

The factors that affect the impact load are: the type of waterway, the flood conditions, the type and draught of vessels and the type of structure (JCR-Ispra 2012).

9.5 Modelling example of Milau Viaduct using CSI Bridge

The Millau Viaduct was designed by Foster + Partners. It is a multi-span cable-stayed structure that has been claimed to be currently the tallest bridge in the world. It consists of eight spans; six central spans of 342 m each and two end spans of 204 m. In total, it has a complete length of 2460 m. The pylons are set into the deck in both the longitudinal and transverse direction. This is to ensure continuity between the metal sheets of the central box girder and those of the walls of the pylon legs and also to provide rigidity through a frame that covers the bearings found on each pier shaft. The deck is in the form of a trapezoidal profiled steel girder box with a depth of 4.20 m with an orthotropic decking made up of metal sheets.

Two models are set up based on the architectural drawings provided by Foster + Partners (as shown in Figure 9.6). One is a 3D SAP2000 model of Millau Viaduct (as is demonstrated in Figure 9.7). Similar modelling techniques will be introduced in Section 9.6 to model the Forth Bridge in Scotland. Here, in order to demonstrate how to model a bridge in CSI Bridge, the model will be set up in CSI Bridge and for demonstration purposes only two spans are modelled. The detailed steps will be introduced here.

9.5.1 Model set up

- Launch CSI Bridge as shown in Figure 9.8, click on *New Model*. The model will be constructed from a blank model and the units are set to KN, m.

Fig. 9.6   Architectural drawings provided by Foster + Partners.
Image courtesy of Foster + Partners.

Fig. 9.7   Model of Millau Viaduct in SAP2000.
SAP2000 screenshot reprinted with permission of CSI.

Fig. 9.8   Launching CSI Bridge.
SAP2000 screenshot reprinted with permission of CSI.
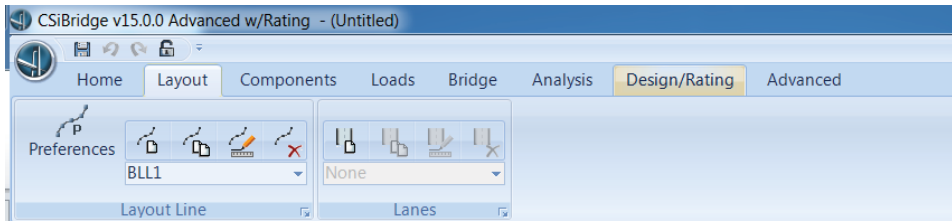
Fig. 9.9   Layout module.
SAP2000 screenshot reprinted with permission of CSI.
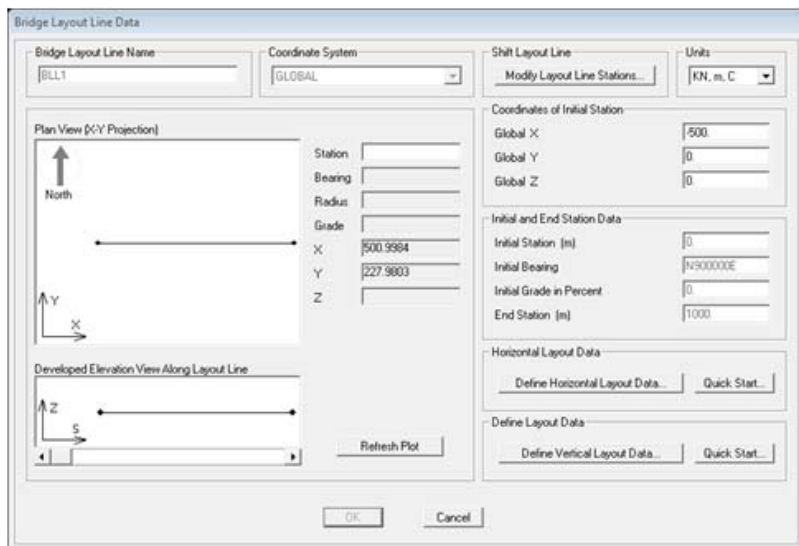
Fig. 9.10   Defining the bridge layout.
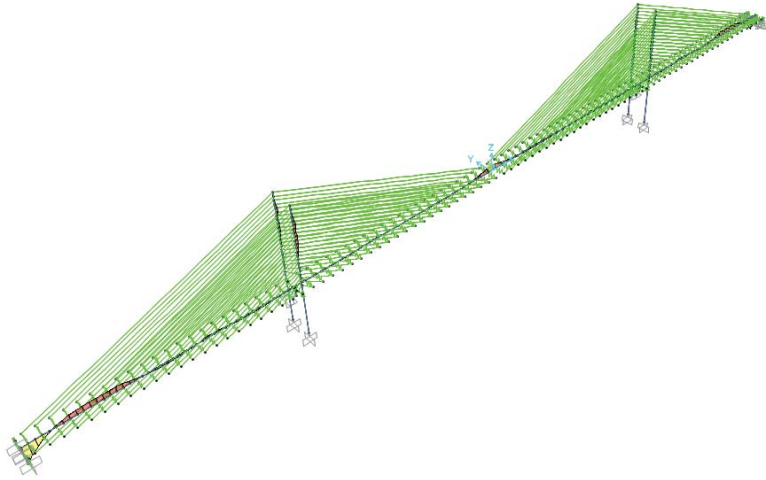**SAP2000 screenshot reprinted with permission of CSI.**

Define the layout

- Go to *Layout Line* – click on *Add New Layout Line* as shown in Figure 9.9.
- A new window will pop up (Figure 9.10). The layout of the bridge will be defined; this is to set the origin of the bridge in order to establish a coordinate system. Defining the origin is mandatory, as when inserting bridge components at a later stage users must insert objects using coordinates and they are also used to set dimensions.
- Selecting the *Components* tab (Figure 9.11), then go to the *Type Menu* and set frame properties for the towers. Similar to SAP2000, the material and section type can be created within this operation. For this example, a uniform section has been selected; however, if a non-uniform section needed to be employed, this would need to be done via the non-prismatic section menu. The objects would need to be created separately and then combined using the non-prismatic menu.
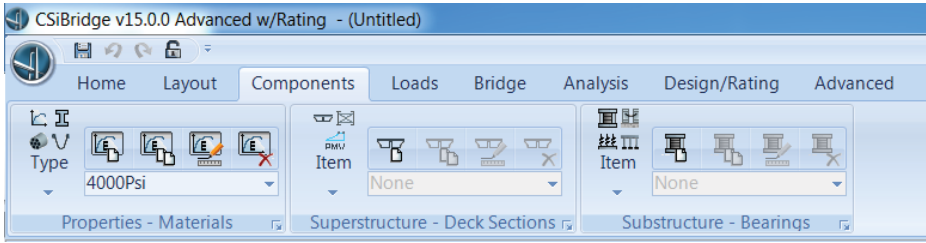
Fig. 9.11   Bridge component module.
SAP2000 screenshot reprinted with permission of CSI.
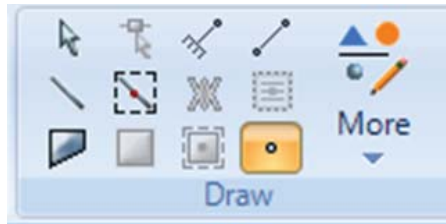
Fig. 9.12   Selecting frame elements from draw module.
SAP2000 screenshot reprinted with permission of CSI.

Define the tower

- Switch to 'XY' view at Y = 0. Extra features will have to be used when drawing the towers as they are not standard components.
- Selecting the draw *Frame* (Figure 9.12) command and setting the properties object box to *Pylon* (pre-defined name), the user can draw the Tower and edit the height and location of each respective point using the 'joint coordinates' editor.
- Joints are created along the tower to provide for the cable connections. One joint is created at a specified location by the user. Other joints can then be simply added by using the replicate command.

Define the deck sections

- Select the *Components* tab. A new deck is selected from the menu (Figure 9.13); various types are available in different materials. For the cable-stayed bridge, steel girders were selected. The software allows the user to input the dimensions of the deck.

Cable connections

- The cable connections located at the deck must be defined; go to the *Bridge* module, click on the *Span Items* tab as shown Figure 9.14.
- The user points command is then used to divide the deck into the required segments as shown Figure 9.15.
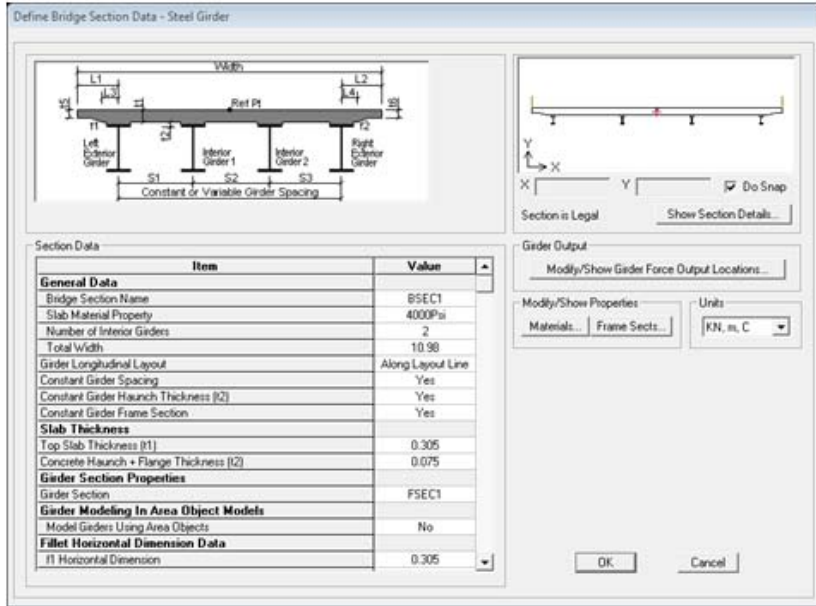
Fig. 9.13   Deck section menu.
SAP2000 screenshot reprinted with permission of CSI.

Fig. 9.14   Bridge module.
SAP2000 screenshot reprinted with permission of CSI.
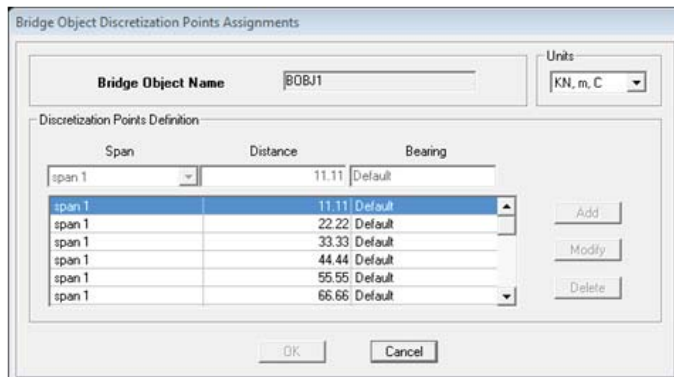
Fig. 9.15   Cable location points.
SAP2000 screenshot reprinted with permission of CSI.
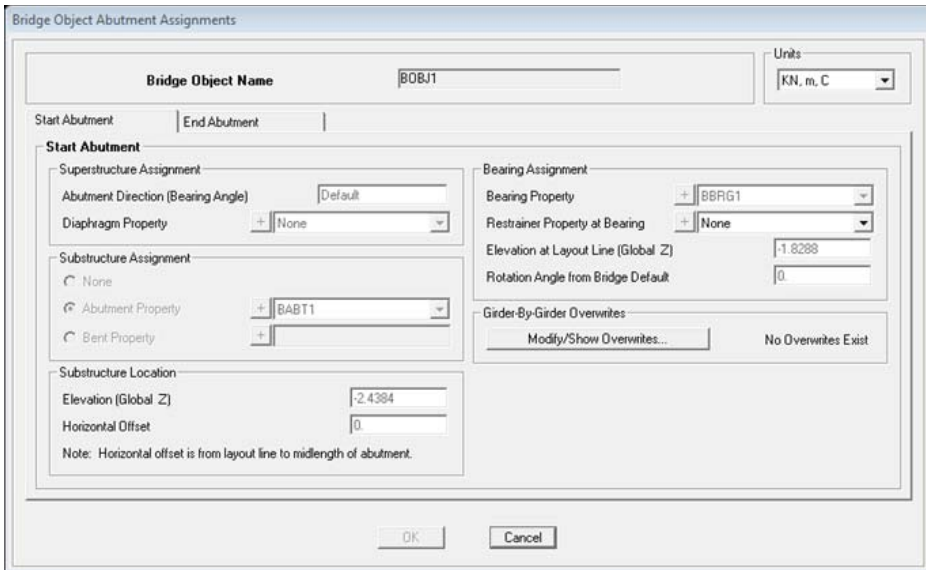
Fig. 9.16 Abutment definition menu.
SAP2000 screenshot reprinted with permission of CSI.

9.6 Defining abutments

- CSI Bridge provides a catalogue of different abutment types. Go to the *Bridge* tab and click on the *Supports* menu. Bearing can be modified within the abutments to allow for movements in the direction required, as shown in Figure 9.16.

Define the rigid links

- As the locations of the cables have been defined along the deck and tower, rigid links will now be placed in these positions. These will form the connection between the deck and cable. Users can define restrictions on movements depending on which movements are preferred.
- This option can be found in the *Advanced* tab and *Rigid Link* menu.

Defining the cables

- CSI Bridge allows users to define cables either by inputting a cross-sectional area or a diameter. Material properties can also be altered to replicate the steel that will be employed on site.
- This option can be found under the *Components* tab by selecting *Cable Properties*. When drawing the cables, the user must start from the joint of the tower and join to the required rigid link. It is important to take care as the cable could be connected to the wrong link.
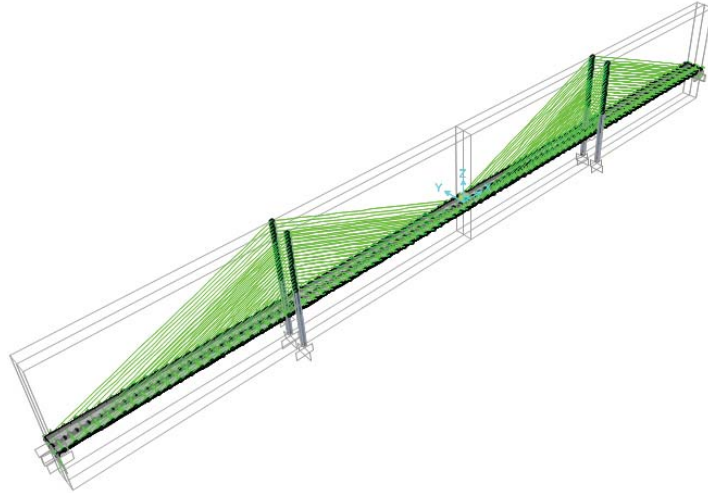
Fig. 9.17 3D bridge model in CSI Bridge.
SAP2000 screenshot reprinted with permission of CSI.
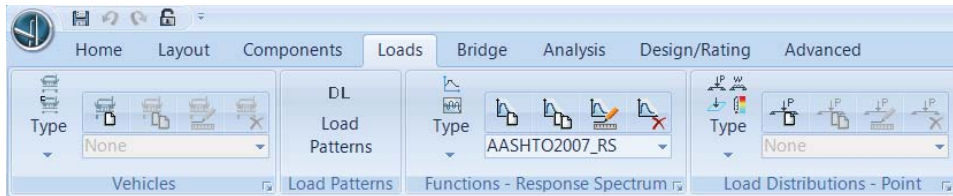
Fig. 9.18 Loads module.
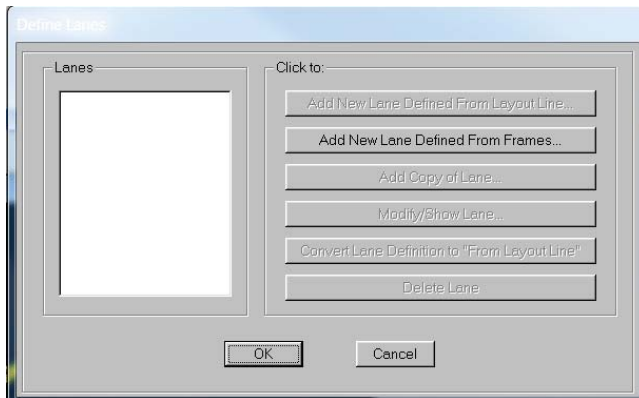**SAP2000 screenshot reprinted with permission of CSI.**
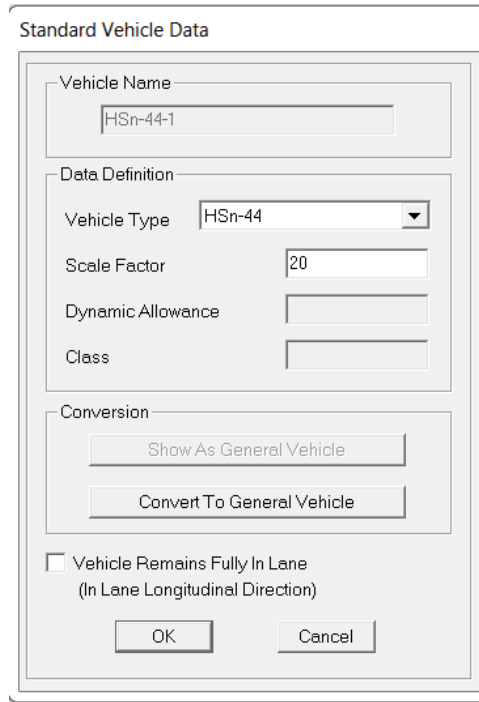
Final model

Following the above steps, the model has been set up as shown Figure 9.17.

9.6.1 Define the vehicle loading
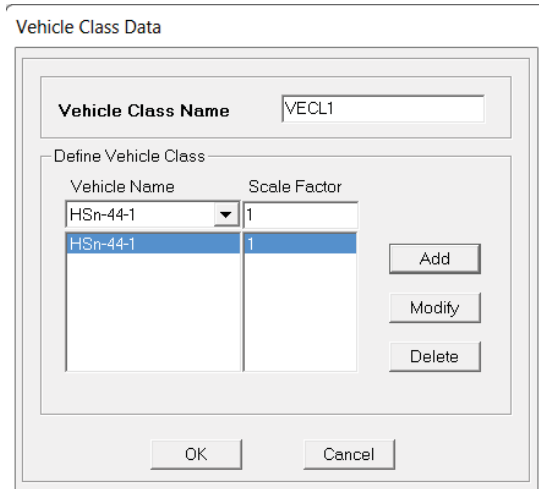
In bridge design, vehicle loading is one of the most important considerations; in this section, we are going to introduce how to define vehicle loading in CSI Bridge.

1. Click on the *Loads* module (Figure 9.18).
    - Go to *Types*; under the *Vehicle* tab click on *Vehicles – Add A New Vehicle*; a new window will pop up as shown in Figure 9.19.
    - Choose vehicle type *HSn-44*, which is a truck.
    - Go to *Types* again; click on *Vehicles Classes – Add A New Vehicle Class*; a new window will pop up as shown in Figure 9.20.
    - Add a new vehicle class with the name *VECL1*, choose the vehicle we just defined, click *Add* (Figure 9.20).

Fig. 9.19   Selecting vehicle type.
SAP2000 screenshot reprinted with permission of CSI.

Fig. 9.20   Selecting vehicle class.
SAP2000 screenshot reprinted with permission of CSI.
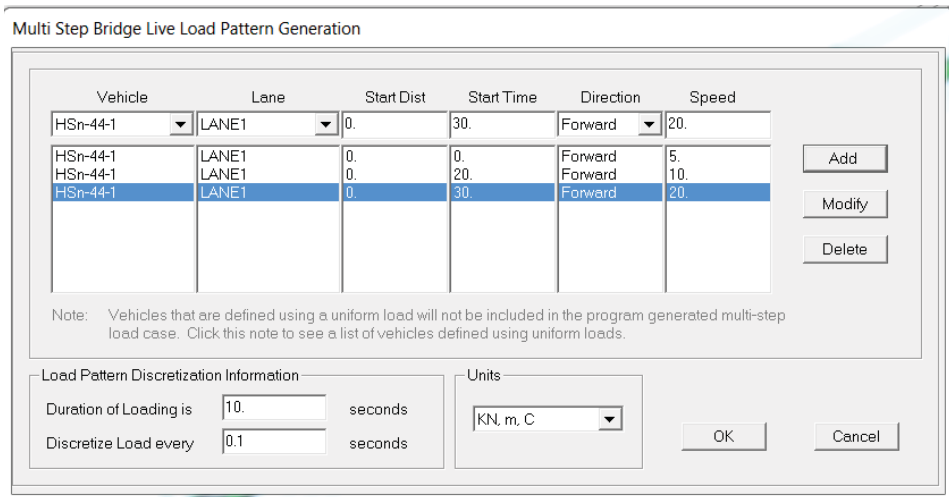
Fig. 9.21   Defining the bridge live load pattern.
SAP2000 screenshot reprinted with permission of CSI.
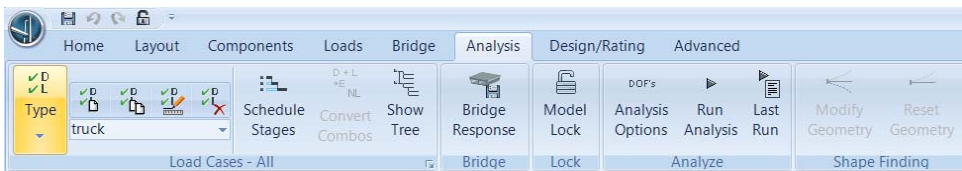
Fig. 9.22   Analysis module.
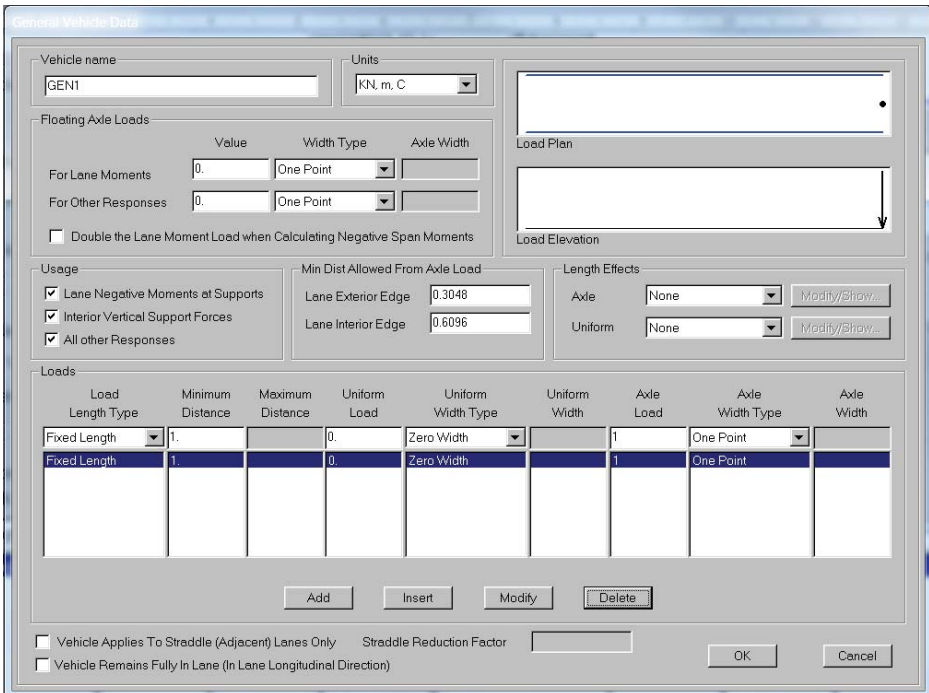SAP2000 screenshot reprinted with permission of CSI.

- Click on *Load Patterns*; add a new *Load Pattern Class* with the name *Traffic*, type *Bridge Live*, add three vehicle durations with different starting times and different speeds as shown in Figure 9.21.

2. Click on *Analysis Module* (Figure 9.22).
   - Go to *Load Cases* tab – click on *Add A New Load Case*; a new window will pop up as shown in Figure 9.23.
   - A new window will pop up as shown in Figure 9.24.
   - Give the name *Traffic*.
   - Change the *Load Case Type* into *Time History*.
   - Add *Load Pattern*, choose the pattern we just defined.
   - Set up the remaining parameters as shown in Figure 9.24.
3. Run *Analysis*.

9.6.2   Analysis and result

After analysis, the results such as bending moment, shear force and deflection can be checked. Figure 9.25 gives an example of the analysis results.
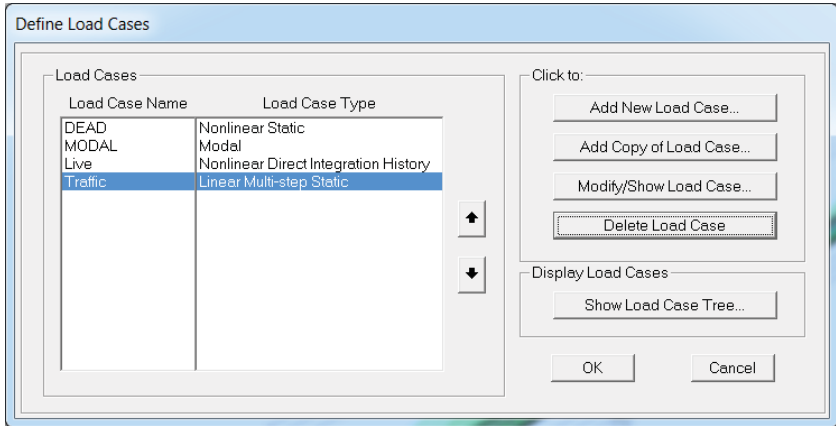
Fig. 9.23 Creating the traffic load case.
SAP2000 screenshot reprinted with permission of CSI.
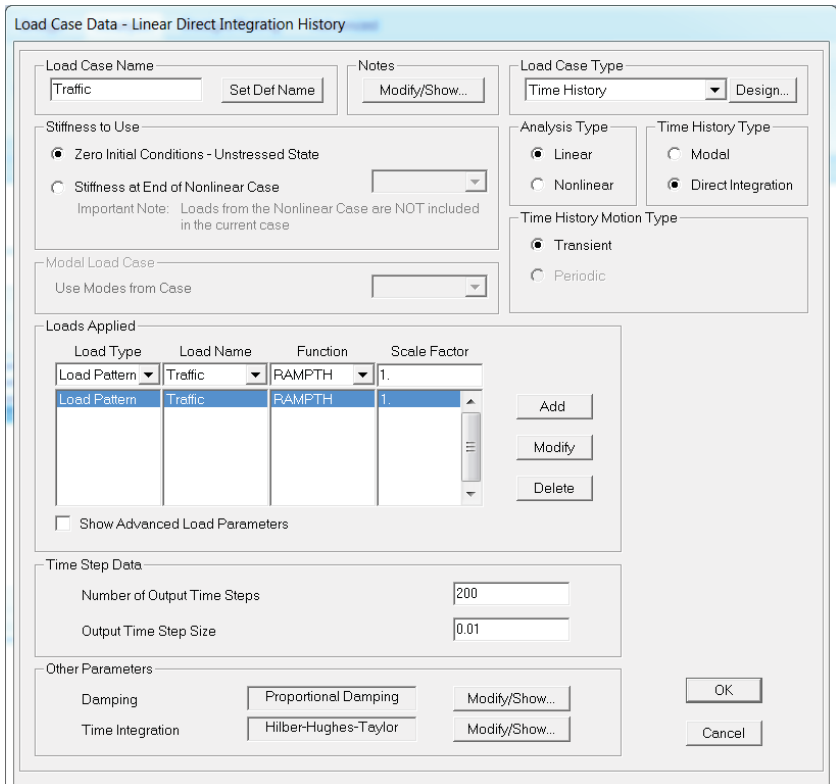
Fig. 9.24 Defining the traffic load case parameters.
SAP2000 screenshot reprinted with permission of CSI.

Fig. 9.25   Bending moment diagram.
SAP2000 screenshot reprinted with permission of CSI.

## 9.7   Modelling example of Forth Bridge using SAP2000

For more complicated bridge geometry, SAP2000 is another good program for modelling. For lower versions of SAP2000, if a license for the Bridge module is purchased, you can perform the same analysis as in CSI Bridge. Here, we are going to use the Forth Bridge in Scotland as a prototype to demonstrate how to model it in SAP2000.

The Forth Rail Bridge was built in 1890. The primary objective of this bridge is to connect the two coasts of Scotland. The structural engineering and other construction methodologies used in this bridge are still amazing in current world engineering and technology.

Sir Benjamin Baker came up with a design for a suspended cantilever bridge made up of six cantilever spans held in place by three towers and two spans in the middle to connect the cantilever spans to the towers. To connect this whole structure to either side of the coast, two approach viaducts were constructed with a number of abutments to fix the ends of the cantilever at both ends. The foundations were sunk at identical intervals across the whole width to achieve symmetry for the whole structure. The bridge behaves similarly to a typical truss, where the members are in compression and tension.

In this model, the member size and properties were taken from the introduction on the web and member dimensions were scaled according to photos on the web.

Step 1: Set up a 3D model in AutoCAD

The first step is to set up AutoCAD 3D-wireframe model and import it to SAP2000 as a .DXF file.
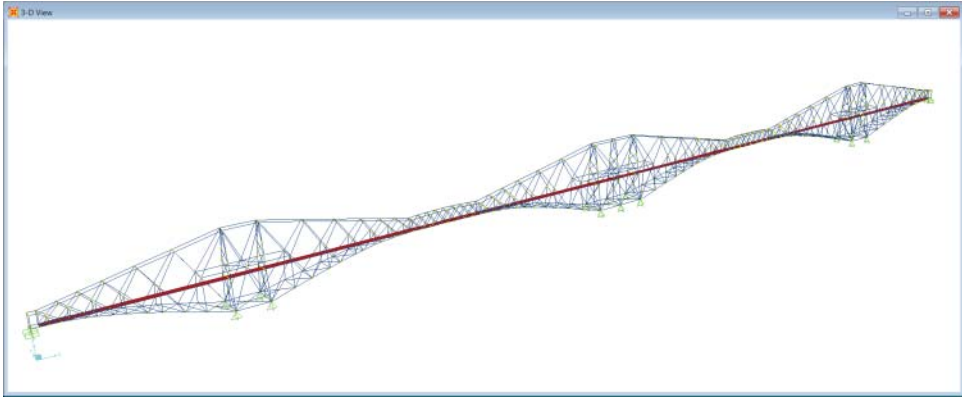
Step 2: Import 3D models in AutoCAD into SAP2000

Similar to that mentioned in other chapters, after importing the structural frame sequentially from different layers of AutoCAD, it is convenient to place the members from each layer into the same group. This will facilitate the future modification of the member.

Figure 9.26 shows the model after importing into SAP2000.

Step 3: Defining member properties and member size

* Go to *Define – Section Properties – Frame Sections*.

Each member material property was assumed to be Fy50 steel from reference, and this is defined with the properties shown in Figure 9.27.

The bridge was modelled using both box and tubular sections. The design check was run through using the design module in SAP2000.

Step 4: Defining the deck

The deck of the bridge consists of an internal viaduct with three rail tracks. The deck is modelled with a thin shell member (as shown in Figure 9.28) with a thickness of 250 mm, which is an approximated dimension. Viaduct or individual track members are ignored; however, their weights have been added as dead load in the analysis.

Step 5: Defining moving load cases

In the bridge design, the moving load of the vehicle is an important consideration. Here we are going to demonstrate how to model it in SAP2000. The other load cases, such as dead and other types of live load, will be defined in the same way as introduced in other modelling examples.

* Go to *Define* – click on *Bridge Loads, Lanes* as shown in Figure 9.29.
* A new window will pop up as shown in Figure 9.30.
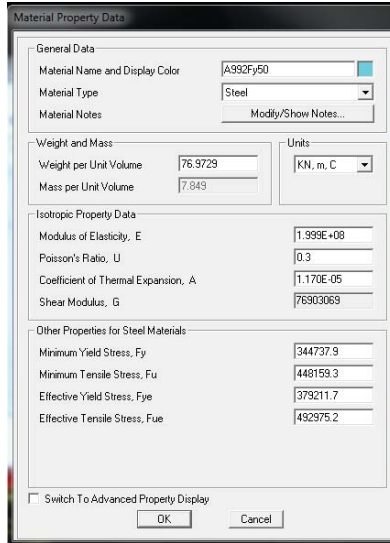
Fig. 9.27   Member material properties.
SAP2000 screenshot reprinted with permission of CSI.
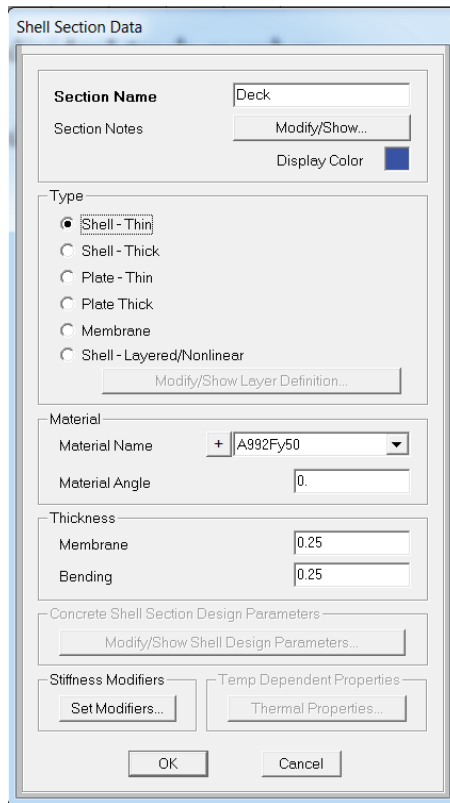
Fig. 9.28   Defining the shell element for deck.
SAP2000 screenshot reprinted with permission of CSI.
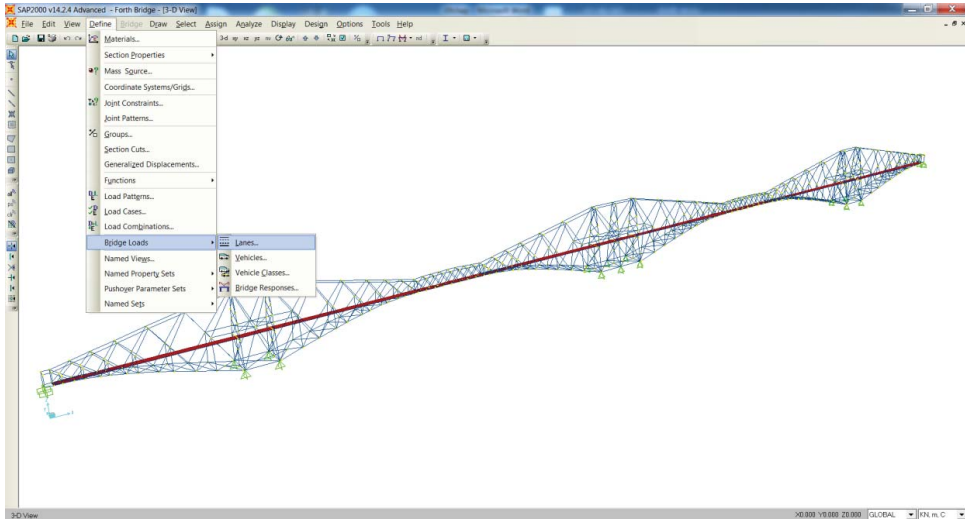
Fig. 9.29   Selecting the bridge loads module.
SAP2000 screenshot reprinted with permission of CSI.

Fig. 9.30   Define the lanes.
SAP2000 screenshot reprinted with permission of CSI.

- Click on *Add New Lane Defined From Frames*.
- A new window will pop up (Figure 9.31).
- Go to *Set Display Options*, switch on the *Frame/Cable/Tendons – Labels*, the frame number will be shown in Figure 9.32.
- Put in the ID number of the frame element where you want to define the lanes. Here, the display colour is green, so after the definition the frame elements chosen will be highlighted green (as shown in Figure 9.32).
- Go to *Define* –click on *Bridge Loads – Vehicles*.
- A new window will pop up (Figure 9.33).
- Click on *Add General Vehicle*.
- A new window will pop up; add the new vehicle load as shown in Figure 9.34.
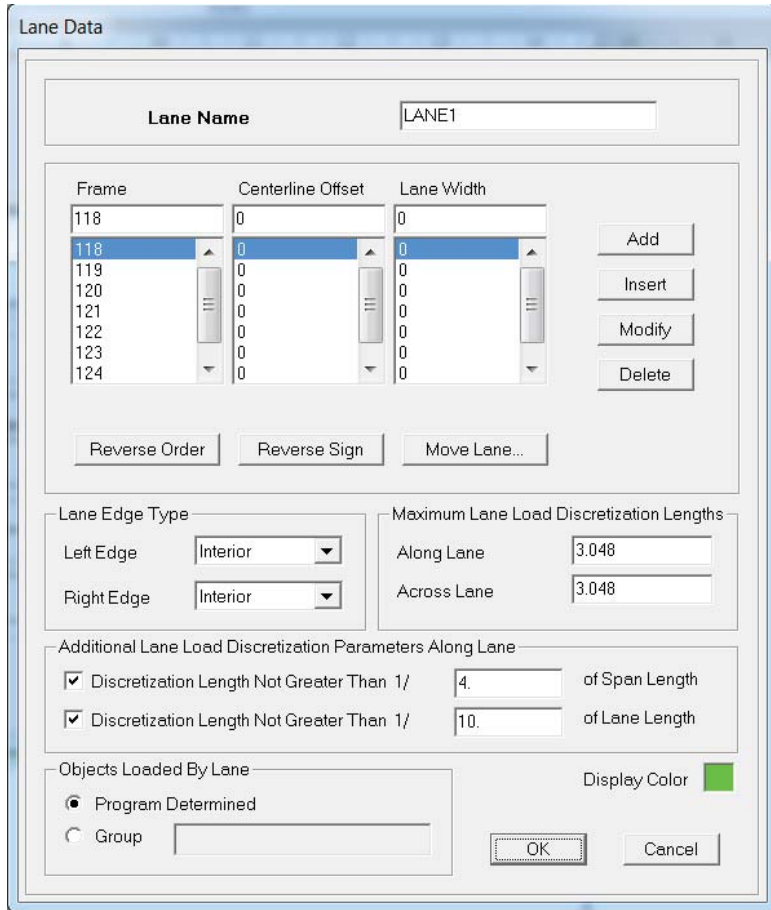
Fig. 9.31   Setting the lane parameters.
SAP2000 screenshot reprinted with permission of CSI.
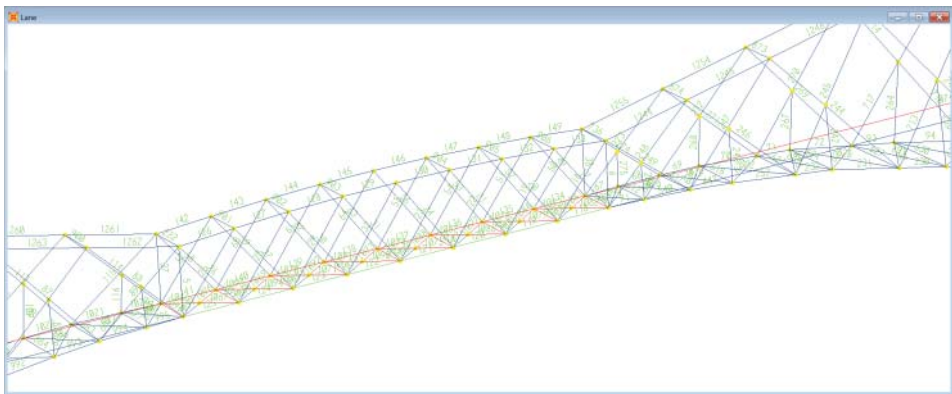
Fig. 9.32   Frame elements highlighted in green.
SAP2000 screenshot reprinted with permission of CSI.
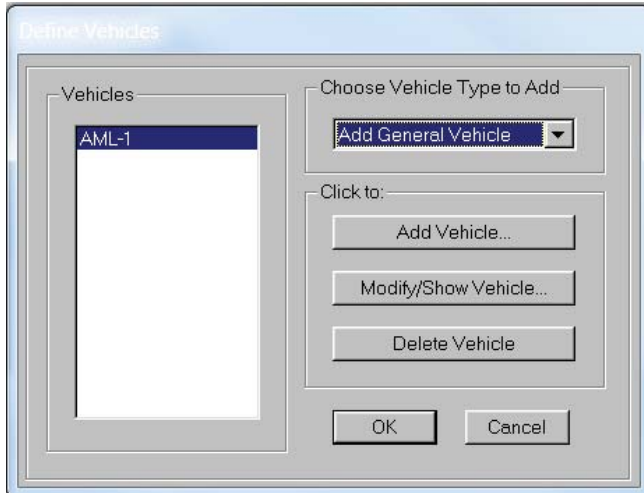
Fig. 9.33 **Adding general vehicle.**
SAP2000 screenshot reprinted with permission of CSI.

Fig. 9.34 **Adding the new vehicle load.**
SAP2000 screenshot reprinted with permission of CSI.
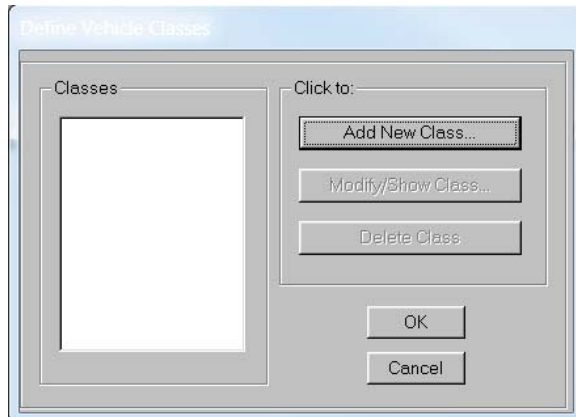
Fig. 9.35   Adding the vehicle class.
SAP2000 screenshot reprinted with permission of CSI.
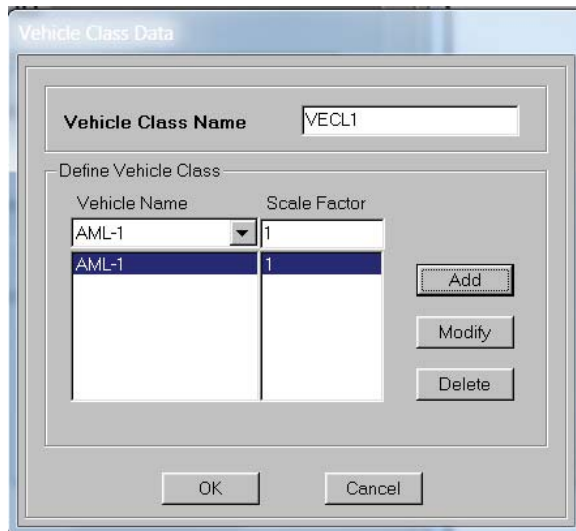
Fig. 9.36   Selecting the vehicle class.
SAP2000 screenshot reprinted with permission of CSI.

- Go to *Define* – click on *Bridge Loads – Vehicles Classes*; a new window will pop up (Figure 9.35).
- Click on *Add New Class*, a new window will pop up as shown in Figure 9.36, add the new vehicle classed as shown below.
- Go to *Define – Load Cases – Add New Load Cases*, a new window will pop up as shown Figure 9.37; add the moving load as below.

Make sure you provide the time history data for the *Moving Load* analysis.
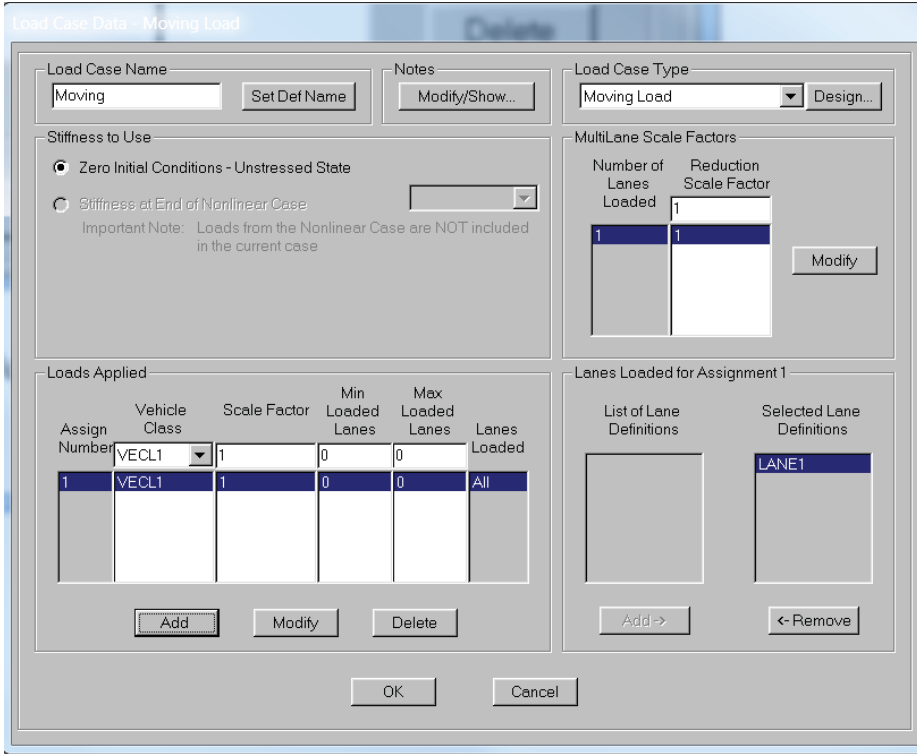
Fig. 9.37   Defining the moving load case.
SAP2000 screenshot reprinted with permission of CSI.
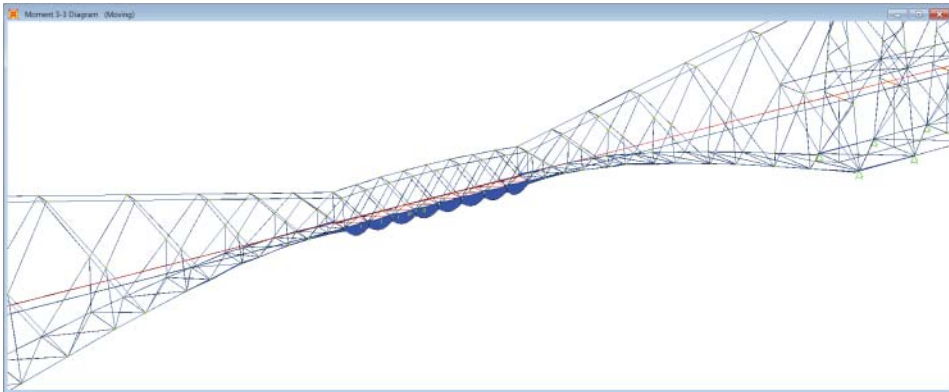
Fig. 9.38   Result of bending moment due to moving load.
SAP2000 screenshot reprinted with permission of CSI.
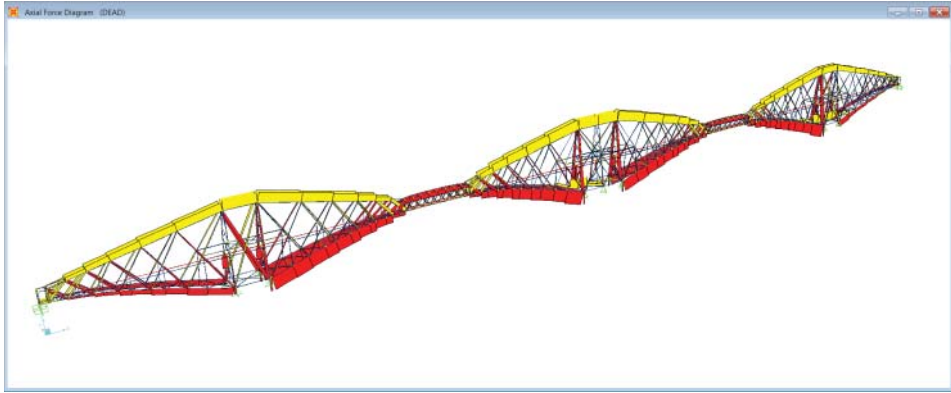
Fig. 9.39   Result of axial force due to dead load.
SAP2000 screenshot reprinted with permission of CSI.

Step 6: Analysis and result

After the setting up, the model is analysed and the result can be interpreted. Figure 9.38 shows the bending moment under the moving load case at the centre of the bridge.

Figure 9.39 shows the axial force under the dead load case of the bridge.

References

Billah, K., and Scanlan. R. 1991. "Resonance, Tacoma Narrows Bridge failure, and undergraduate Physics Textbooks" *American Journal of Physics* 59, 2: 118–124.

Nowak, A. 1993. "Live load model for highway bridges," *Structural Safety*, 13: 53–66.

JRC-Ispra. 2012. "Seminar on 'Bridge Design with Eurocodes'", organized and supported by European Commission, Russian Federation, European Committee for Standardization.

Codes and regulations

AASHTO. 2007. Standard specifications for highway bridges, American Association of State Highway and Transportation Officials, Washington, DC.

EN1991-2 2003 Eurocode 1: Actions on structures - Part 2: Traffic loads on bridges, European Committee for Standardization.

10 Foot-induced vibration

10.1 Introduction to vibration problems in structural design

In current design practice, vibration is another important issue that needs to be addressed. Walking, running and jumping on floor slabs or footbridges will produce a dynamic force that can induce appreciable vibrations. The vibrations can cause discomfort to residents and pedestrians and sometimes it will also cause deterioration of the structural integrity. Particularly in some special types of building, such as hospitals and laboratories, vibration problems should be carefully addressed in the design, as these types of buildings are more sensitive to vibrations. For instance, vibration will affect an operation in a ward of a hospital. It will also affect the accuracy of a test result in the lab. Therefore, for a structural engineer, how to design a structure to resist vibration is another important skill.

10.2 Characteristics of foot-induced dynamic loads

The foot-induced dynamic load is affected by factors such as walking speed, length of step and pacing frequency. In the past, most studies have concentrated on human-induced vertical forces and vibrations. However, the horizontal component of foot force is also worth investigating; one of the most famous examples is the problem observed on the Millennium Bridge in London as a result of human loads. A modelling example using SAP2000 of the Millennium Bridge problems due to human-induced vibration will be presented in the later part of this chapter.

10.2.1 Pace frequency

Several experiments on foot-induced vibration have confirmed that the general vertical forcing frequency values of a pedestrian walking are between 1.4 Hz and 2.4 Hz. Among them, Matsumoto et al. (1972) took a sample of 505 individuals, and concluded a normal distribution in the pacing frequency with a mean of 2.0 Hz and a standard deviation of 0.173 Hz.

The lateral forcing frequency actually lies between 0.7 and 1.2 Hz, as the lateral component of the force is applied at half the footfall frequency; this will be explained in the later part of this chapter.

Natural vertical and lateral frequencies, as mentioned, are within the limits of many footbridges and therefore have the potential to cause excessive vibrations of the structure. This explains the need to consider both horizontal and vertical pedestrian loads acting on a structure.

10.2.2   *Vertical loading*

In the past, studies have been carried out where more attention has been paid to finding the vertical component of the dynamic force. Figure 10.1 illustrates the form of vertical loads upon a structure, caused by an individual taking one step, from the studies of Bachmann (2002) and Wheeler (1982).

Measurements of individuals walking have also been taken (Bachmann, 2002). The time history of individuals walking indicated a near periodic output, with an average step frequency equal to an average period. Assuming a perfect periodicity of the force, Figure 10.2 represents the general shapes for continuous forces in the vertical direction.

10.2.3   *Horizontal loads*

Figure 10.3 shows the general shapes for continuous forces in the horizontal direction.

Horizontal loads on a structure have not been well quantified in the past, more research need to be done. Lateral oscillations of the structure are the result of movements that occur due to the human's natural walking pattern, using their right and
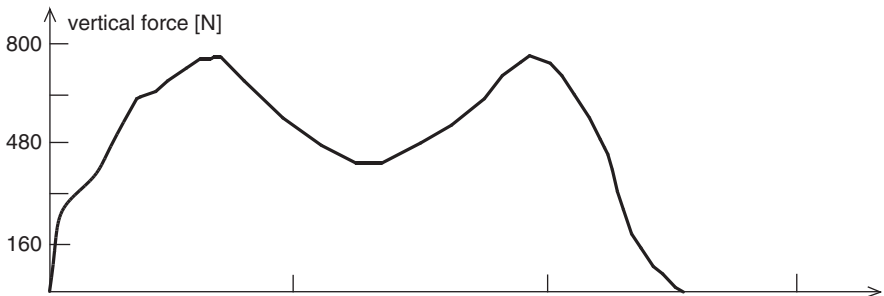
Fig. 10.1   Vertical loads from one step (reproduced based on data in Bachmann 2002 and Wheeler 1982).
AutoCAD.
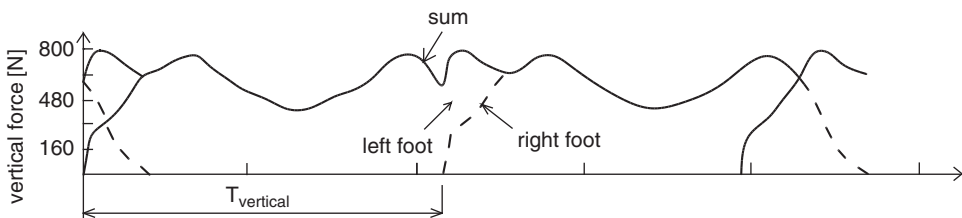
Fig. 10.2   General shapes for continuous vertical forces (reproduced based on data in Bachmann 2002).
AutoCAD.
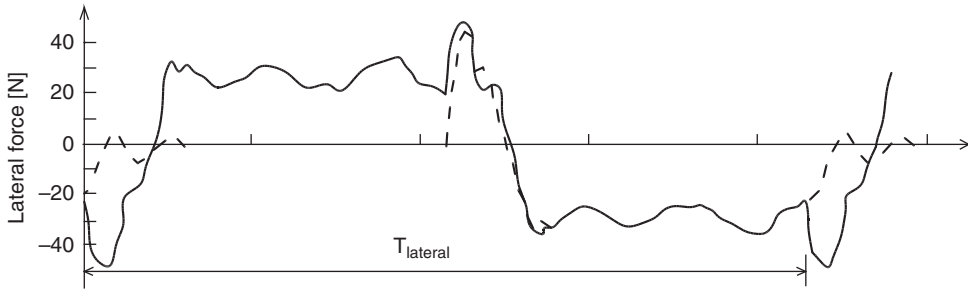
Fig. 10.3   Periodic walking time histories in the horizontal direction (reproduced based on data in Bachmann 2002).
AutoCAD.

left foot in turn. These lateral oscillations produce amplitudes in the range of 1–2 cm. Lateral oscillations of the centre of gravity of a structure such as the footbridge, as well as the lateral oscillations due to body movement when walking on a structure, are what produce the horizontal dynamic forces on these structures.

The human pacing frequency is twice the frequency of the horizontal force, hence approximately between 1.4 Hz and 2.4 Hz, at a horizontal force of between 0.7 Hz and 1.2 Hz. It was found that on a stationary surface, the frequency of the horizontal force is around 10% of the vertical load, which happens to be 4% of the individual pedestrian weight (Bachmann, 2002).

10.2.4   Loads induced by groups and crowds

The problem of excessive vibration of footbridges has occurred as a result of crowds and groups of people walking on a structure at a given time. Humans interact with the structure as a result of excessive vibrations, resulting in human–structure synchronisation, where the pedestrians start to develop and adapt their step sequence to the vibrations of the structure. This happens with footbridges, where vibrations of the bridge due to increased vertical and horizontal loading cause a synchronising effect with the way humans move on the structure. Human–bridge synchronisation happens both in vertical and horizontal directions, with both having an impact on the excessive movement of footbridges. It results in a combination of horizontal and vertical loads.

10.3   Acceptance criteria

Generally, vibration is a serviceability issue related to human discomfort; however, it is hard to quantify as the perception of discomfort varies between humans. In current design practice, the perceived acceleration of the floor or deck is used to define the acceptance criteria.

Humans are less likely to feel discomfort when a structure vibrates vertically than if it vibrates horizontally. This is mainly due to the fact that vertical vibration is more frequent than horizontal vibrations (as discussed in the earlier section).

Table 10.1 Maximum allowable acceleration according to EN1990 (2002), A2.4.3.2.

Direction & Type	Maximum acceleration m/s²
Vertical vibrations	0.7
Horizontal vibrations, normal use	0.2
Horizontal vibrations, crowd conditions	0.4

Permission to reproduce this extract granted by BSI.

Low frequency lateral motion on the surface of a structure like a footbridge causes discomfort to pedestrians. This is mainly due to horizontal synchronisation, which will cause horizontal motion of the footbridge, and at times this synchronisation may develop into a motion similar to that of the small horizontal vibrations of the bridge, resulting in excessive vibration of the structure and causing serious discomfort and safety issues.

For the Millennium Bridge in London, human–structure synchronisation due to lateral loading resulted in excessive movement of the structure (Ove Arup, 2001).

10.3.1   Footbridge

In serviceability design, EN1990 (2002) uses the maximum allowable accelerations of any section of the deck in defining the comfort of pedestrians. The suggested values for maximum accelerations, according to EN1990 (2002) A2.4.3.2, are presented in Table 10.1.

EN1990 (2002) also recommends that dynamic loads induced by pedestrians with a frequency equal to one of the fundamental natural frequencies of the footbridge should be identified for a vibration limit state validation.

10.3.2   Floor slabs

Floor slabs are prone to vertical vibrations, due to their large in-plan stiffness, the lateral vibration problem is hard to induce. BS6472 (2008) defines the acceptance criteria for floor slabs in terms of multiples of the base acceleration curve. In Figure 10.4 the base acceleration curve is in the z-axis, there are also curves available for the x and y axes. The base curve is derived based on the RMS (Root Mean Square) acceleration rather than peak acceleration. The base RMS acceleration in the vertical direction in the range of $0.005\,\text{ms}^{-2}$. These criteria vary with frequency, with the most onerous requirements in the fundamental frequency range 5 to 8 Hz; this can be seen from Figure 10.4.

One thing we need to note is that these values are for continuous vibration, whereas footfall vibration tends to be intermittent in practice. Recognising the intermittent nature of the loading, the vibration dose value of the acceleration is suggested by BS6472, 2008 and ISO10137, 2007. This calculation is based on an estimate of the amount of time in any, usually 24 hour, period that the intermittent response will take place. The RMS response seen under any period of loading can then be adjusted by a factor that is a function of the percentage of time that the loading takes place. The
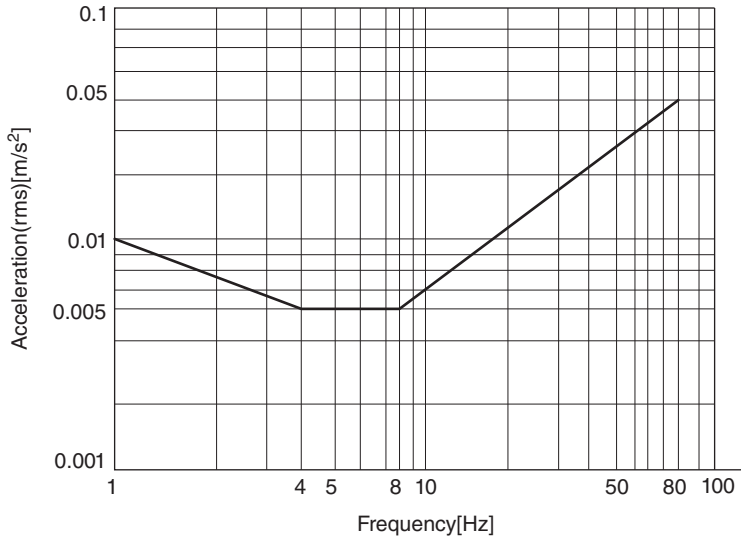
Fig. 10.4   Base acceleration curve (z-axis) to ISO 10137(2007).
Permission to reproduce this extract granted by BSI.

vibration dose value can be worked out using the below equation:

$$VDV = \left(\int_0^T a_w(t)^4 dt \right)^{1/4}$$ (10.1)

Where:
VDV is the vibration dose value (m/s$^{1.75}$)
$a_w(t)$ is the weighted acceleration (m/s^2)
T is the total period of the day during which vibration may occur
a_{rms} is the root mean square acceleration

For hospital floor vibration design, vibration acceptance criteria for ward and theatre areas in hospital are contained within HTM2045 (1996). The above method is acceptable under HTM 2045 (1996) for ward areas but is not allowed for operating theatres where the response criteria are the most onerous. This guidance recommends response factors of 1.0 for theatre areas and 1.4 for ward areas at night. The response factors are worked out by multiples of the base curves for acceleration outlined in ISO 10137 (2007) and are shown in Figure 10.4.

In Figure 10.4, the base curves in the vertical direction (z-axis) are derived from the base value that

$$a_{rms} = 5 \times 10^{-3} m/s^2$$ (10.2)

Where:
a_{rms} is the root mean square acceleration

10.4   Loading representation of foot-induced vibration

The following section will concentrate on the mathematical modelling of pedestrian-induced loads, which will allow us to solve the equation of motion. Similar to other dynamic analysis such as earthquake and blast analysis, the structure can be idealised as mass (m), damping (c) and stiffness (k) system, subjected to the external foot-induced dynamic loading $F(t)$. Simplifications and assumptions will be made in both the representation of the structure and the loading.

As stated in Chapter 4, the governing equation of motion of floor systems or footbridges can be stated as:

$$m\ddot{x} + c\dot{x} + kx = F(t) \tag{10.3}$$

Where:
$m\ddot{x}$ is the inertial force
$c\dot{x}$ is the damping force, and c is the damping coefficient
kx is the elastic force
$F(t)$ is the external dynamic force (here, it is induced by foot)

$x, \dot{x}, \ddot{x}$ are the displacement, velocity, acceleration of the structure respectively solution of the equation of motion can be carried out either in the time domain or the frequency domain. The choice of solution technique depends on the nature of the loading and the response.

Time-based solutions can be carried out directly (linear or nonlinear) by solving the governing equation of motion or via conventional modal superposition techniques (if it is a linear structure), which requires eigenvalue analysis. For linear structures, modal superposition techniques are preferred for time-based analysis as they are much more computationally efficient, provided sufficient modes are captured.

Frequency-domain solutions are based on random response analysis, which is also based upon modal superposition. Frequency-domain solutions are generally more computationally efficient than time history analysis.

10.4.1   Time-domain solution (time history analysis)

The time-domain solution method represents the foot-induced dynamic load in terms of a forcing function on a time basis. It is based on the assumption that both feet falls produce the same period force to the floor, which is simply transposed along the time axis at the period of walking to derive a full time history. In reality, the position that this force is applied on a structure varies in space as the person applying the force walks across the structure. However, the assumption is conservative. In vibration analysis, the force is placed in positions to invoke the worst case response of the structure. In addition, the fundamental walking frequency should be chosen such that one of the higher harmonics of the period force coincides with the fundamental frequency of the floor system to make the worst case response.

The most common mathematical representation of the period force in the vibration is to define the loading as a Fourier series as outlined below.

$$f_p(t) = Q + \sum_{n=1}^{K} Q\alpha_n \sin(2\pi nft + \varnothing_n) \tag{10.4}$$

Where:
Q   is the pedestrian weight
f is the frequency of the force
n is the harmonic number
k is the total of the harmonics contribution
α_n is the nth harmonic's load factor
$\varnothing_n$ is the nth harmonic's phase angle

The load factor, α_n is essential for this load model and can be determined as the result of different studies such as Bachmann (2002) and Young (2001) for both the vertical and horizontal component.

10.4.2   Frequency-based solutions (random analysis)

A major problem with the time domain representation of loading is that it is too simple to capture the full complexity of the actual load applied to the system, as loading is heavily dependent upon the characteristics of the individual walking on the floor or a footbridge.

A study of Brownjohn et al. (2004) discovered that foot-induced loading has a frequency content that is spread either side of the main harmonics rather than just directly at the frequency of the main harmonics. This effect is thought to significantly contribute to the differences between real response and the response calculated using the artificial time history techniques presented earlier. Particularly for floor structures whose main response is to the higher harmonics of the fundamental walking frequency, the most conservative response will usually occur directly under the point of loading. Therefore, another way to represent human footfall loading is though a random process where the loading is defined in terms of frequency content and the solution of the equations of motion can then take place in the frequency domain.

The random response analysis model of Brownjohn et al. (2004) requires definition of the pedestrian loading in the form of power spectral density (PSD) or auto-spectral density, which depends on the probability distribution of pedestrian pacing rates. The response could be obtained by multiplication of PSD for force. The square root of the response PSD gives an estimate of the RMS response. Such a technique also has the advantage that it can be readily extended to consider the structural response from a large number of forced inputs (as in crowd loading) as long as the coherence between the likely inputs can be determined.

In the analysis, a power spectral density function can be defined based on a small representative sample of walkers given in Brownjohn et al. (2004). Similar to the time domain method, it is also based on a dynamic load factor component for each harmonic and a function defining the spread of load around the harmonic frequency

concerned. The random analysis based on above method can be implemented through programs such as Abaqus® or ANSYS, which will be introduced in Section 10.7.

The frequency domain analysis has a number of advantages over the time history approach:

1. Accurately captures the effects of closely spaced modes which can predominate when there are regular structural bays.
2. Computationally more efficient and gives slightly more conservative results.
3. A complete response map of the floor plate can be plotted, which will enable the engineer to make an effective design.

10.5   Modelling example of vibration analysis for the Millennium Bridge using SAP2000 (time-based method)

In this case study, the Millennium Bridge (as shown in Figure 10.5) will be modelled as a prototype. A time based analysis method will be demonstrated using SAP2000.

On its opening day, it was calculated that a maximum of 2000 pedestrians were on the bridge at any one time, therefore the crowd density ranged between 1.3 to 1.5 pedestrians per square metre. The crowd resulted in horizontal vibration of the Millennium Bridge due to a phenomenon known as synchronised horizontal pedestrian loading, which was explained in the earlier sections.

The majority of the horizontal excitations occurred at the south span of the bridge with a frequency of about 0.8 Hz, and at mid span with a frequency of 0.5 Hz for the

Fig. 10.5   Millennium Bridge.
Photo from Structurae: International Database for Structural Engineering, courtesy of Wilhelm Ernst & Sohn Verlag.
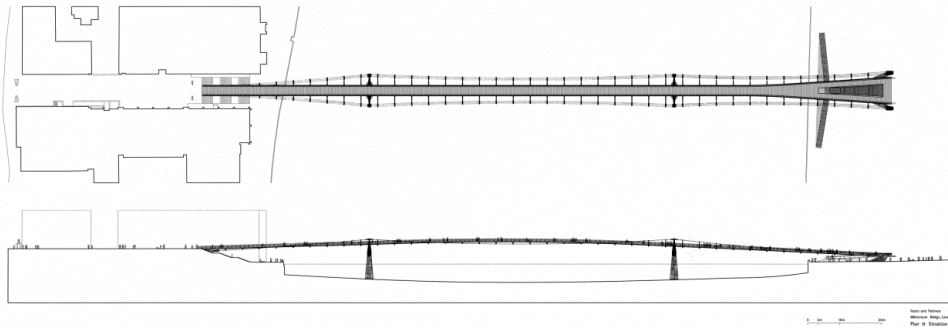
Fig. 10.6   Architectural drawing of Millennium Bridge provided by Forster + Partners. Image courtesy of Foster + Partners.

first lateral mode and 0.9 Hz for the second lateral mode. Analysis indicated that the mid span of the bridge had a deflection of 70 mm, consequently forcing the bridge to close for further investigation and improvements to be made (Ove Arup, 2001).

10.5.1   Model set up

Due to its complexity, the geometry of the Millennium Bridge was created in Auto-CAD and saved as a DXF file, it was then imported to SAP2000 to analyse the effects of pedestrians upon the structure. The structural members, such as the cable plate, were also created in AutoCAD and imported to SAP2000. The model is set up based on some of the architectural drawings provided by Forster + Partners, as shown in Figure 10.6.

The conventional structural members were defined in SAP2000 using the section modules. Members of a more complicated shape, such as the transverse arm which tapers in size, were created using the advanced section designer in SAP2000 (see Figure 10.7).

Properties of materials that were not predefined within SAP2000 were created by adjusting relevant properties such as the modulus of elasticity and weight per unit volume.

The 3D SAP2000 model can be seen in the Figure 10.8.

In the analysis, only the bridge deck was modelled as the dynamic effects imposed by pedestrians only affected the bridge deck. Therefore the bridge piers and substructure were neglected in the analysis and not modelled. Also, the lateral loading effects of wind were ignored due to the insignificance of its effects on lateral vibrations on the opening day of the real Millennium Bridge.

10.5.2   Simulation of pedestrian loads

The effects of vertical and lateral induced accelerations were modelled separately with the target of investigating both phenomena independently.

The modelling of each footstep of an individual pedestrian was generated by applying two point loads at the same position where the footstep acts upon the bridge. At
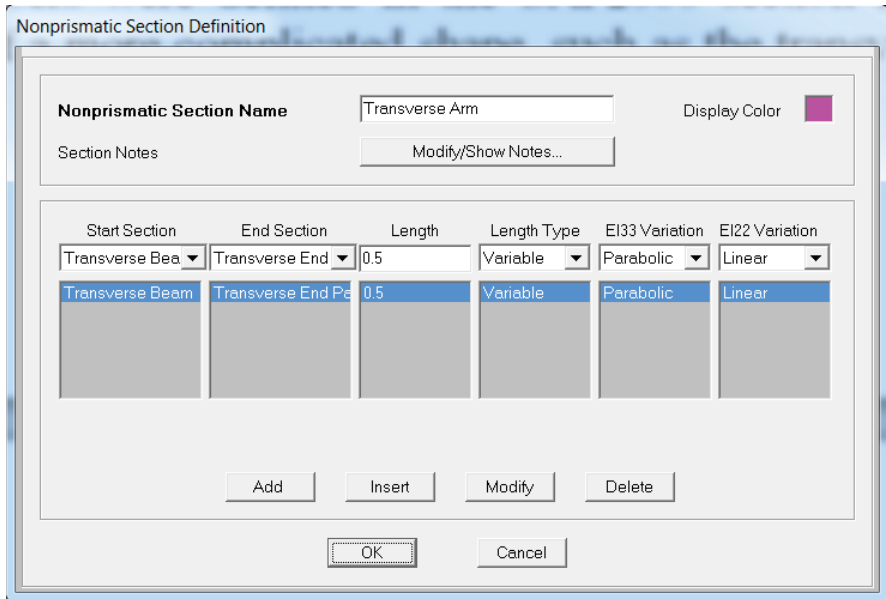
Fig. 10.7   Nonprismatic section definition.
SAP2000 screenshot reprinted with permission of CSI.
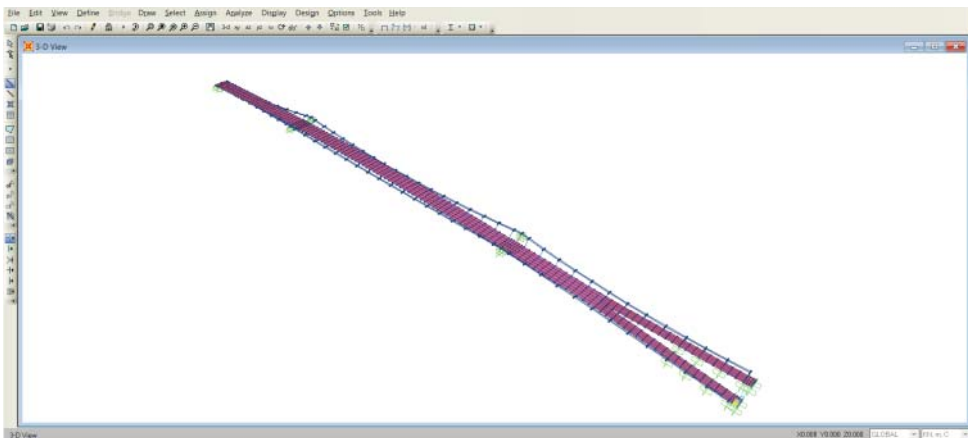
Fig. 10.8   SAP2000 model of Millennium Bridge.
SAP2000 screenshot reprinted with permission of CSI.

the single joint, one point load represented the lateral load (see Figure 10.9, named L1, L2, L3 …) and the other load represented the vertical load (see Figure 10.10, named V1, V2, V3 …) induced by a single step. The point load representing the vertical load was assigned in the global negative Z direction (U3) with a value of 0.75 KN and the lateral load was assigned in the global X direction (U1), with a value of 0.025 KN. These two load values were taken based on a body weight equivalent to 75 kg and the corresponding lateral load produced as discussed earlier.
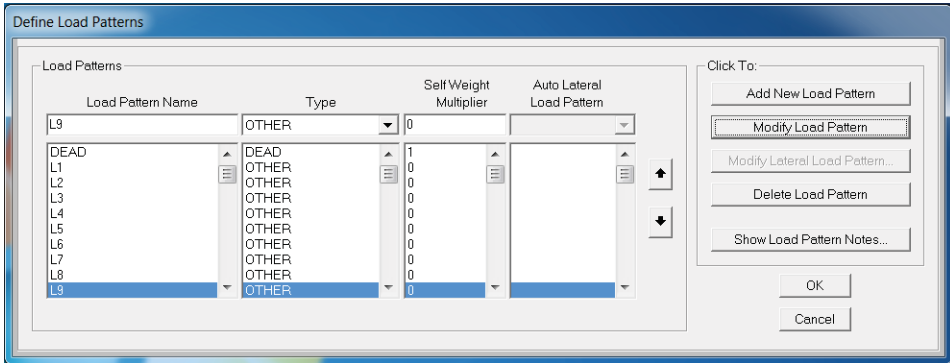
Fig. 10.9   Point load representing vertical component.
SAP2000 screenshot reprinted with permission of CSI.
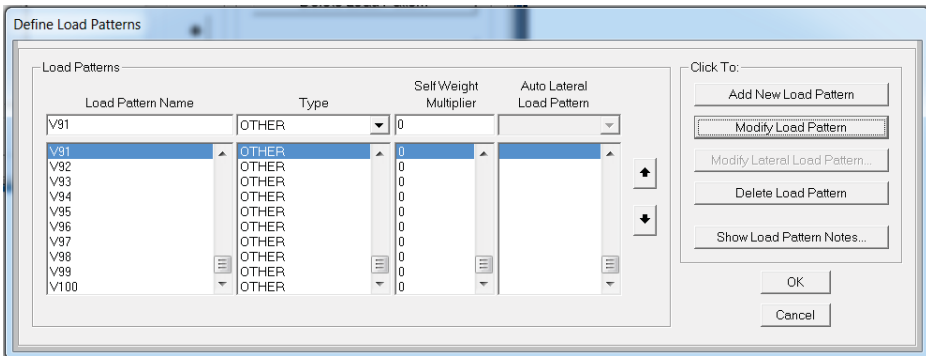
Fig. 10.10   Point load representing horizontal component.
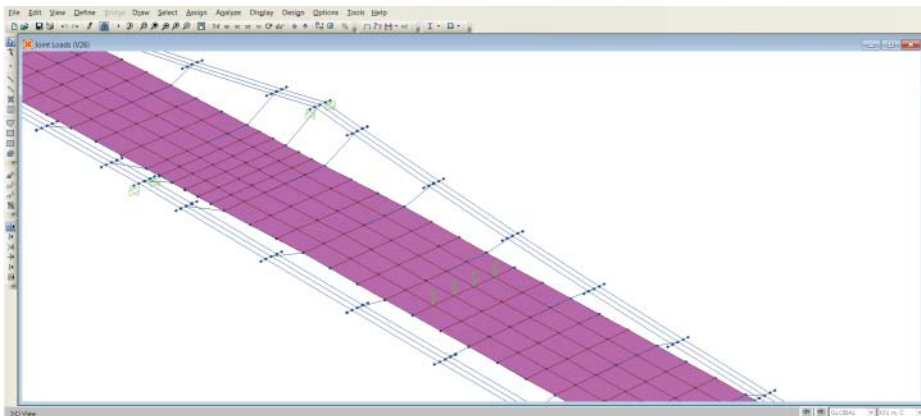
Fig. 10.11   Point load on the bridge deck (vertical direction).
SAP2000 screenshot reprinted with permission of CSI.
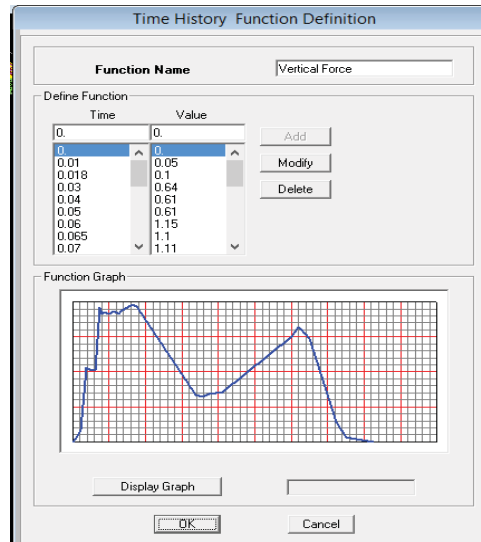
Fig. 10.12   SAP2000 Time history function of vertical force for one step.
SAP2000 screenshot reprinted with permission of CSI.

The above loads were applied at every point along the span of the bridge where a step would fall; as shown in Figure 10.11, in order to apply these point loads, the deck is meshed with the element size approximately equal to the pace of human walking.

All the point loads acting in the horizontal direction or vertical direction were assigned an individual time history load case in SAP2000. For the horizontal loads and vertical point loads, the time history functions were worked out based on the formula mentioned in the earlier section, and they are shown in Figure 10.12 and Figure 10.13.

Each point load case consisted of its own arrival time; this is to simulate people walking along the entire length of the bridge. Multiple point loads with the same arrival time were defined in order to model the effects of synchronisation. Both the lateral load and the vertical load of the same step at the same joint along the deck had the same arrival time as the time at which the loads acting on the bridge are combined.

By carefully positioning the arrival time of each step, an anticlockwise motion of the pedestrians was modelled; this is to simulate the tests conducted by Ove Arup (2001). The effects of approximately 250 pedestrians walking along the bridge were modelled in this exercise.

10.5.3   Analysis of Millennium Bridge before retrofit

Vertical loads induced by pedestrians

The analysis of this model was done firstly under its own dead load as well as the vertical induced vibrations imposed by pedestrians through a time history analysis. (Readers can refer to other chapters for an introduction on how to perform time history analysis, therefore detail on the method is not given here.)
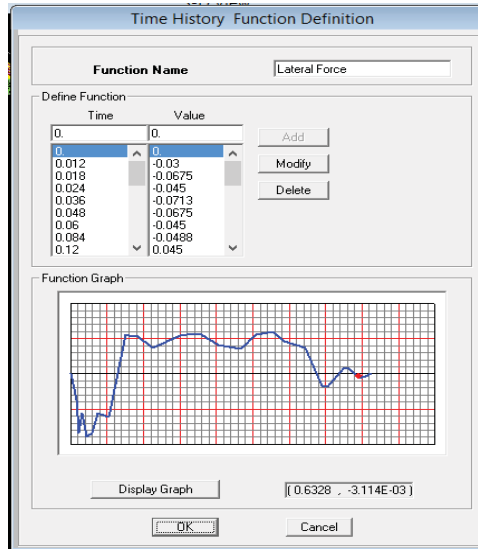
Fig. 10.13   SAP2000 Time history function of horizontal force for one step.
SAP2000 screenshot reprinted with permission of CSI.

After analysis, a time history record was made in SAP2000 to assess the accelerations in a vertical direction. The location on the bridge with the largest acceleration was determined first and a time history trace for a certain points within this location was examined, the detailed steps are shown below:

- Following the same procedures we introduced in Chapter 4, we can plot the time history due to foot force. Click *Display* in the ribbon, choose *Show Plot Function*, put in the joint number you want to investigate, and then click *Display*.

The time history plot window will pop up, as shown in Figure 10.14; the maximum acceleration experienced at joint 1003 is 0.4 m/s^2, which is considerably lower than the
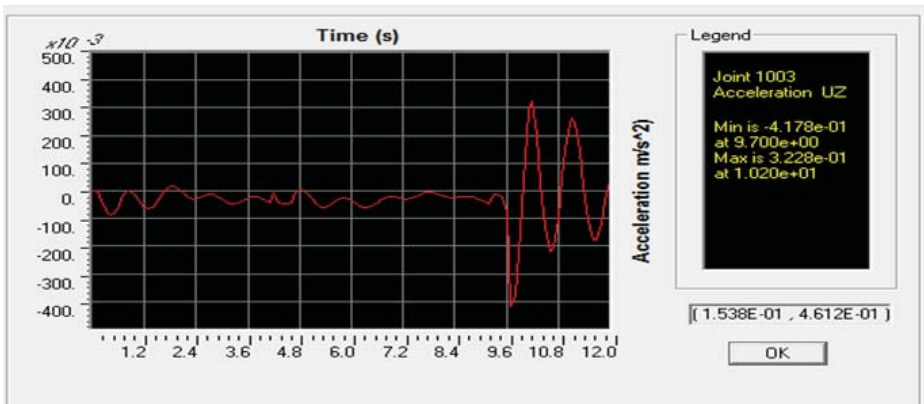
Fig. 10.14   Time history for max vertical acceleration.
SAP2000 screenshot reprinted with permission of CSI.
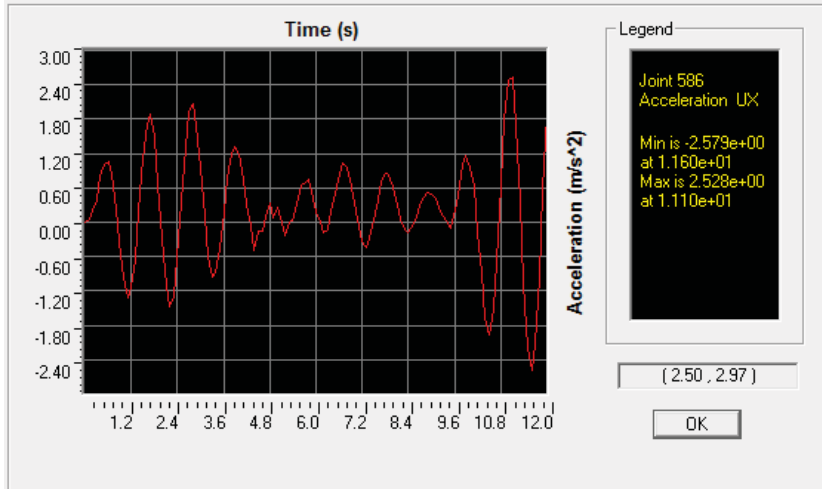
Fig. 10.15   Time history trace for max horizontal acceleration.
SAP2000 screenshot reprinted with permission of CSI.

maximum allowable acceleration according to Eurocodes EN1990 (2002) – which is found to be 0.7 m/s^2. Therefore, vertical loading induced by pedestrians does not have a great impact upon the oscillations of the structure.

Horizontal loading induced by pedestrians

The same model was tested under the horizontal loads induced by pedestrians. The time history record was observed at joint 586 with the maximum acceleration.

It can be noted from Figure 10.15 that the maximum lateral acceleration on the deck was approximately 2.6 m/s^2. In reality, the accelerations experienced by the actual bridge were between 1.9 m/s^2 and 2.45 m/s^2; this value appears to be correct. The allowable horizontal accelerations in Eurocode EN1990 (2002) for simplified structures are 0.2 m/s^2 and 0.4 m/s^2 for individual and crowd, respectively, therefore it is not acceptable.

10.5.4   Analysis of the Millennium Bridge after retrofit

Due to this excessive acceleration, Ove Arup decided to implement a number of damping devices along the span of the bridge. In order to observe the effectiveness of the retrofit design, a model was created to model the bridge with the damping devices.

Dampers were created using a nonlinear link in SAP2000, which had the relevant stiffness and mass assigned to it and was connected to a rigid joint. The dampers, represented by nonlinear links, can be seen in Figure 10.16. The retrofit also consisted of planned bracing across the structure, where the viscous dampers were installed. The bracings are also modelled as can also be seen in Figure 10.16.

The detailed method will be demonstrated here.
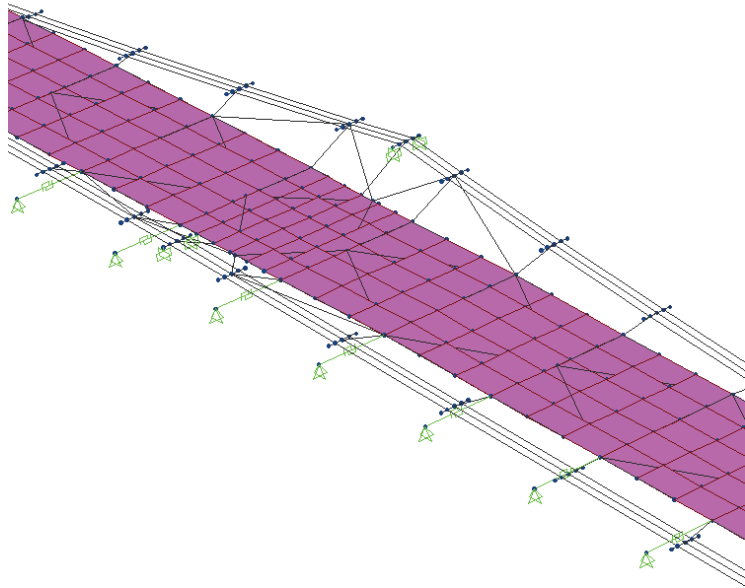
Fig. 10.16   Dampers represented by nonlinear links.
SAP2000 screenshot reprinted with permission of CSI.
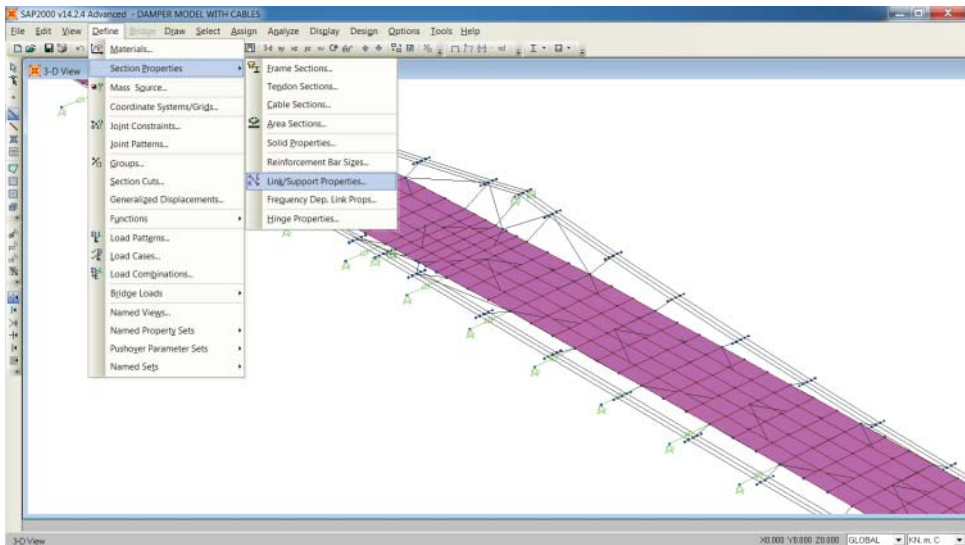
Fig. 10.17   Choosing link/support properties.
SAP2000 screenshot reprinted with permission of CSI.

Modelling the damper

- Click on *Define* from the ribbon, select section *Properties*. Choose *Link/Support Properties* (see Figure 10.17).

- Define a link named *DAMPER* (Figure 10.18), in direction properties, choose *U1* and then click *Modify/Show for U1*.
- A new window will pop up (Figure 10.19), and the property of the damper can be defined here.
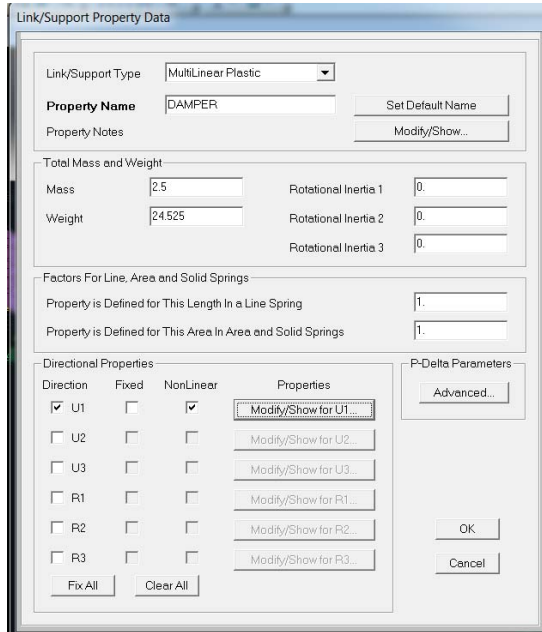
Fig. 10.18   Define link properties.
SAP2000 screenshot reprinted with permission of CSI.
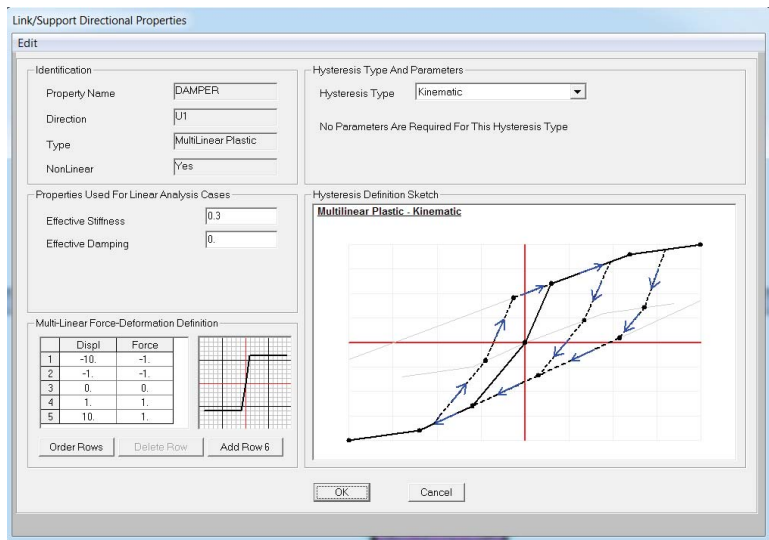
Fig. 10.19   Defining the property of the damper.
SAP2000 screenshot reprinted with permission of CSI.
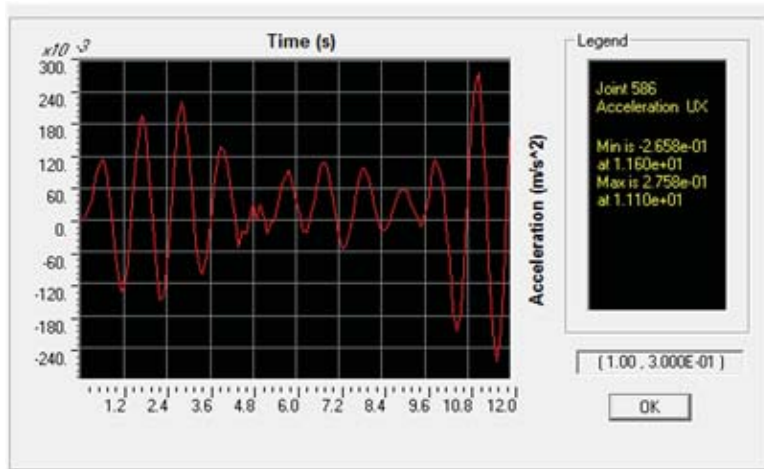
**Fig. 10.20   Time history record for maximum horizontal acceleration.
SAP2000 screenshot reprinted with permission of CSI.**

Analysis and result

The model was tested under horizontal loading induced by pedestrians through a time history load case. The time history trace showing the maximum horizontal acceleration can be seen in Figure 10.20.

The maximum horizontal acceleration of the damped Millennium Bridge model is approximately $0.3 \, \text{m/s}^2$. This is a significant reduction from the un-damped original model which had an acceleration of $2.6 \, \text{m/s}^2$. This shows the effectiveness of the retrofit introduced. Also, an acceleration of $0.3 \, \text{m/s}^2$ is acceptable in accordance with the only design code, EN1990 (2002), which publishes a value for maximum horizontal acceleration of $0.4 \, \text{m/s}^2$.

10.6   Modelling example of vibration analysis of hospital floor using Abaqus® (frequency-based method)

In this case study, a vibration analysis using the frequency-based method will be demonstrated. A floor plate of a hospital building was selected for the analysis. The response of theatre and ward areas to human-induced vibration is investigated. In the vibration analysis, the load case including the power spectral density function is applied at the point that would invoke the worst case response in the structure. This would result in the most conservative response of the system concerned.

10.6.1   Prototype structure

The structure selected for the case study is a predominantly reinforced concrete flat slab structure. The main lateral stability of the structure is provided by reinforced concrete stair cores and shear walls. The model can be seen in Figure 10.21.
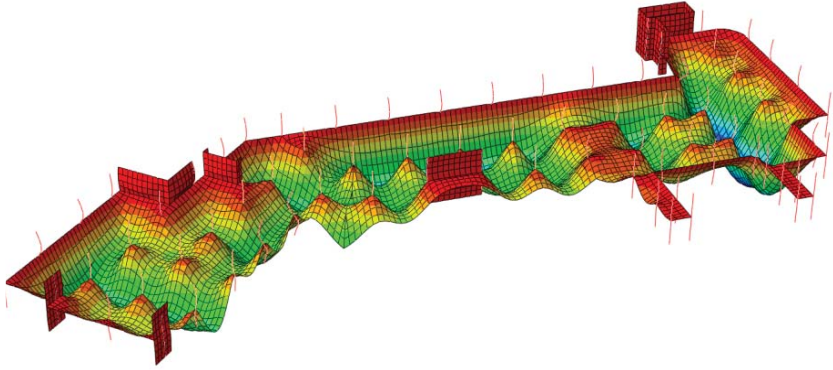
Fig. 10.21   Deformation under static loading (the deformation is amplified).
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

10.6.2   Modelling technique

As mentioned earlier, vibration is mainly a serviceability design issue, therefore it is reasonable to assume linear elastic properties of the structure, although concrete cracks may develop due to vibration. This is required for modal superposition techniques that are linear perturbation processes.

Element selection

Concrete slabs are modelled using 2D shell elements. It is usual to base the Young's modulus of concrete on the higher dynamic value (usually between 35 and 40 kN/mm^2). Columns are modelled using 1D beam elements. For concrete flat slab structures, it is usual to assume that columns are rigidly connected to slab elements, therefore no moment release needs to be defined in the Abaqus® model.

Mesh selection and mass representation

The mesh density is selected as able to represent the main structural modes. Non-structural mass such as lightweight partition walls can be included by the use of added mass elements or by enhancing the density of structural components. They are not included as structural elements although they can contribute to overall damping. As small vibrations are assumed, it is usual to assume that the façade provides sufficient stiffness to provide full vertical restraint to the slab edge in the analysis. In addition, an allowance was made for 10% of the imposed load to be added as the mass. An allowance was also made for the weight of the non-structural screed. Cladding mass was not included as the edge of the slab was vertically restrained.

Material model

Concrete cracking is critical to the vibration performance. Therefore, it is modelled in Abaqus® using a concrete cracking model. The tension stiffening behaviour of the

concrete cracking model is based upon the ICE 372 (Scott, 1983) approach modified for long-term creep.

Damping

The vibration response is very sensitive to the level of damping assumed in the structure. The most common damping value used for concrete structures is 3%. This is the value adopted for all floors in the recent comparative study for the Concrete Centre (2004).

10.6.3   Analysis procedures and major Abaqus® commands used in the simulation

The analysis is performed using the INP file; the 3D model is first set up using the ETABS program and then converted into an INP file in Abaqus® using a program designed by the author (Fu, 2009). Readers can manually set up the files as shown in this section.

There are two INP files that need to be generated; the first analysis is to work out the natural frequency of the floor plate, where an initial static load step is run to determine the cracked state of the slab to enable the extraction of the mode shapes based on the 'cracked' stiffness of the slab.

The next stage is to perform a random analysis of the structure. The second analysis restarts from the first analysis, which means that when you run the second analysis, you need to input the file name of the first analysis.

Modelling steps

Below are the major modelling steps:

- Conduct a static load case to determine the cracked concrete areas of the floor plate and conduct an eigenvalue analysis to extract the main vertical modes of the 'cracked' floor system with mode shapes normalised for generalised mass.
- Determine the worst case modal displacement for each structural bay within the frequency range of interest.
- Work out the power spectral density function of loading for frequency of loading for each structural bay and carry out random response analysis.
- Determine root mean square vertical accelerations for each load case.
- Determine the maximum envelope of vertical accelerations and hence response factors for the whole floor plate and compare against acceptance criteria by the code of practice.

First analysis: work out the natural frequency

The INP file consists of several main parts, the reader can refer to the Abaqus® manual for detailed examples; here only the most important parts are explained in detail.

Part 1. Coordinates (the coordinates of all the nodes are defined here)
*node, nset=Node
1,-41.67209,132.2793,3.65
2,-25.65457,133.6938,0
3,-25.65457,133.6938,3.65
4,-25.65457,133.6938,7.3
5,-16.6542,133.6938,0
6,-16.6542,133.6938,3.65
......
Part 2. Column element (concrete columns are defined here)
*element,type=b31,elset=COL400X4001
1,3,8265
*beam section,section=RECT,elset=COL400X4001,material=concrete
0.40,0.40
-1,0,0
....
Part 3. Core wall element (core walls are defined here)
*element,type=s4r,elset=W225
100001,315,8889,8949,4449
....
Part 4. Slab element (slabs are defined here)
*element,type=s4r,elset=SLAB350
100393,3,419,418,417
....
Part 5. Section properties (here the concrete material and the slab mesh are defined)
*shell section,elset=slab350,material=C1,density=250 (slab is defined)
0.35,9
*rebar layer
t20topx,3.14e-4,0.200,0.15,s460,,1
t20topy,3.14e-4,0.200,0.15,s460,,2
t20botx,3.14e-4,0.200,-0.15,s460,,1
t20boty,3.14e-4,0.200,-0.15,s460,,2
Part 5 core wall section properties
*shell section,elset=w225,material=C2 (wall is defined here)
0.225,5
Part 6. Material
*material,name=C1 (concrete C1 material is defined here)
*Concrete
  4e+07,   0.
 4.5e+07, 0.0025
*Failure Ratios
 1.16, 0.056, 1.28, 0.33
*Tension Stiffening
   1.,   0.
 0.01, 0.0025
*Density
2400.,
*Elastic
 3.8e+10, 0.2

```
*material,name=concrete(concrete C2 material is defined here)
*elastic
38000.0e6,0.2,   0
*density
2400
*material,name=s460(steel rebar material will be defined)
*elastic
205000e6,0.3
```

Part 7. Support and boundary conditions (please refer to Abaqus® manual for further information)

```
*nset,nset=topnode
9672
9668
......
*nset,nset=bottom node
9637
9633
......
*nset,nset=slabedge
917
920
......
*boundary
bottomnode,1,6
topnode,1,2
slabedge,3,3
```

Part 8. Step 1 (static step)

```
*Step, name=Static
*Static
0.25, 1., 1e-05, 1.
*Dload(Define Pressure)
slab350, P, -5500.
*Dload(Define Gravity load)
all, GRAV, 9.81, 0., 0., -1.
*Restart, write, frequency=1(OUTPUT REQUESTS)
*Output, field, variable=PRESELECT(Define FIELD OUTPUT: F-Output-1)
*Output, history, variable=PRESELECT(Define HISTORY OUTPUT: H-Output-1)
*End Step
```

Part 9. Step 2 (In this step the natural frequency will be extracted)

```
*step
*frequency,eigensolver=lanczos,normalization=mass
100,
*output,field
*node output
u
rf
*restart,write,FREQ=1
*end step
```

Figure 10.21 shows the deformation of the floor under the first static analysis step.
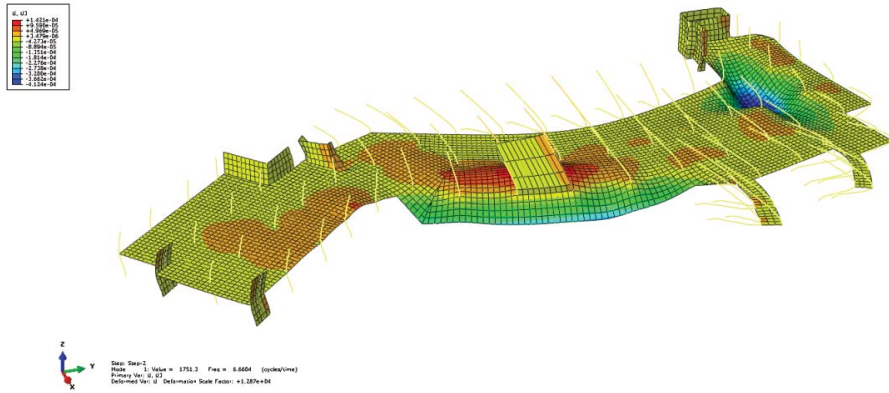
Fig. 10.22   Mode shape of Mode 1 (the deformation is amplified).
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
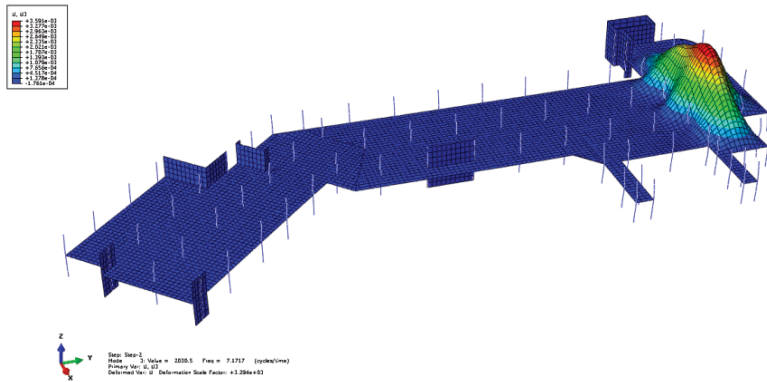
Fig. 10.23   Mode shape of Mode 3 (the deformation is amplified).
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Part of the modal analysis results are shown in Figure 10.22, Figure 10.23 and Figure 10.24, which represent the results of Mode 1, Mode 3 and Mode 100 respectively.

Random analysis

In this analysis, a new INP file will be generated. The power spectral density (PSD) will be defined. When defining a PSD, a frequency function and a cross-correlation definition (if you want to model the correlation between the vibration caused by a crowd of people, however, this becomes more complicated and therefore is not considered here) need to be specified. The frequency functions are defined as model data (i.e. they are step independent) and must be named. A log scale is used in interpolating between the given values. The default type is power units (Abaqus® 2013).
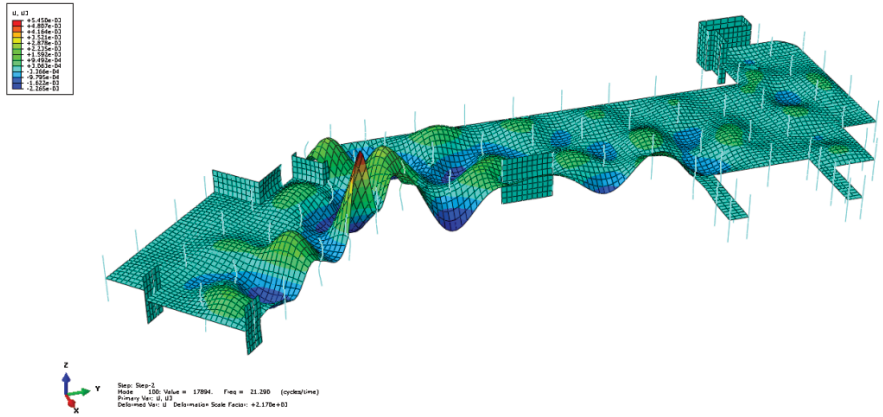
Fig. 10.24   Mode shape of Mode 100 (the deformation is amplified).
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

The method for defining the PSD in Abaqus® is as follows.

*PSD-DEFINITION, NAME=name, TYPE=FORCE (default; power units)

The second INP file used for this modelling example is defined as follows:

Part 1. Restart (here the restart file is the ODB file from the result of the analysis of the first INP file)
*restart, read,step=2

Part 2. Define the PSD load case (the first line is the command to define the PSD, the remaining lines define the amplitude, the last two lines are used to define the location of the foot fall loading, so you can define as many PSD load cases as possible; normally, we will choose the location that is more sensitive to vibration disturbance, such as a ward, operation theatre in the hospital)

*psd-definition, name=foot1,type=force
0.000,,2.06811
4335.098,,2.09011
8123.972,,2.11211
16990.021,,2.13411
40929.080,,2.15611
....
*nset,nset=load1
341
Part 3. Random analysis steps (the num-
ber of steps should be the same as the PSD load case you defined)
*step
*random response
1,20,9,5,1

```
*cload,load case=1
load1,3,-1.0
*select eigenmodes, generate
1,100,1
*modal damping
1,100,0.03
*correlation,type=uncorrelated,psd=foot1
1,1
```

- (Define the Field output):

```
*output,field,frequency=20
*node output
ru
rv
ra
rf
```

- (Define the History output)

```
*output,history,frequency=1
*node output,nset=load1
ru
rv
ra
*end step
```

10.6.4   Analysis result interpretation

In frequency domain analysis, an engineer can investigate the response from each mode summed over the frequency range of interest to estimate total response. Therefore, the response required is the response at the end of the frequency range.

For random response analysis using Abaqus®, root mean square values of acceleration, velocity and displacement can be directly output at pre-selected points. For each model, it is desirable to generate a response map of the whole floor plate, amalgamating response factors from a worst case dynamic loading of the floor plate for each structural bay. For each model, this entails a separate load case for each structural bay of interest.

How to construct the response map is outlined below.

Step 1: create a frame in 'session step' for field output

When the analysis finishes:

- Click *Tool – Create Field Output – From Frames* to make a new step session
- A new window will pop up (Figure 10.25); in *Operation* select *Find the Maximum Value Over All Frames*.
- In *Load Case Name*, put the name of the new session step, here we type in *Max RA*.
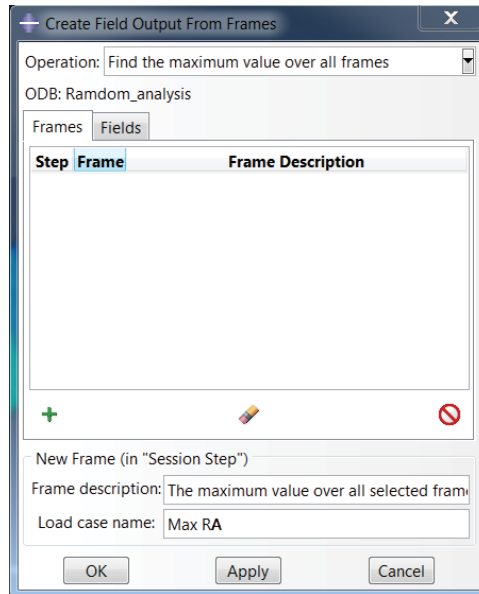
Fig. 10.25   Creating a new frame for the maximum value over all frames.
Abaqus$^{®}$ screenshot reprinted with permission from Dassault Systèmes.

- Click the + symbol.
- A new window will pop up (Figure 10.26). Choose the last frame for each step and click *Apply* until all the steps have been chosen.
- Click *Cancel*, check all steps have been chosen. Insert the name of the frames, for example *Max RA*, click *OK*.

A window will pop up as shown in Figure 10.27.

Step 2: response factor map

Following the explanation of Section 10.3, the base root mean square acceleration (RMA) is 0.005; therefore, the response factor map can be constructed by dividing the RMA by base square mean root acceleration (0.005).

- Click *Tool* in the ribbon, choose *Create Field Output* – select *From Field*.
    A window will pop up as shown in Figure 10.28.
- Change name to *Response factors*.
- Find maximum value of all frames.
- In the *Step* option, choose *Session Step*. In the frame option, choose *Load Case*: *Max RA*. Make sure *Session Step* and *Load Case* are chosen.
- Choose *Root Mean Square Translational Acceleration*; in *Expression*, Type *S33f2_RA_max /0.005*, where 0.005 is the base RMA stated, then click *OK* (Figure 10.29).
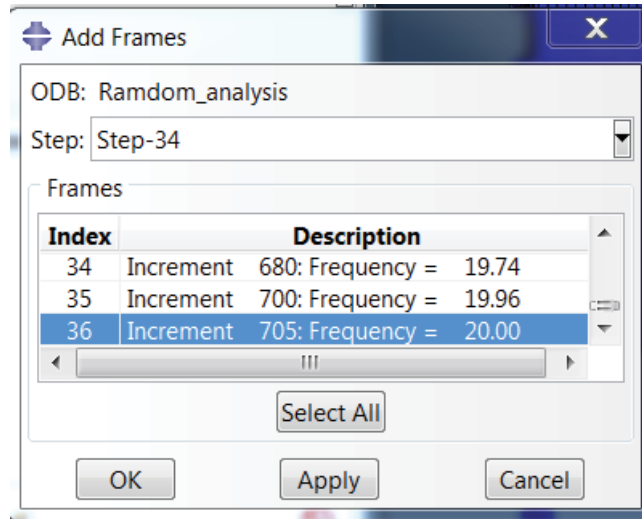
Fig. 10.26   Choosing the last frame for each step.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
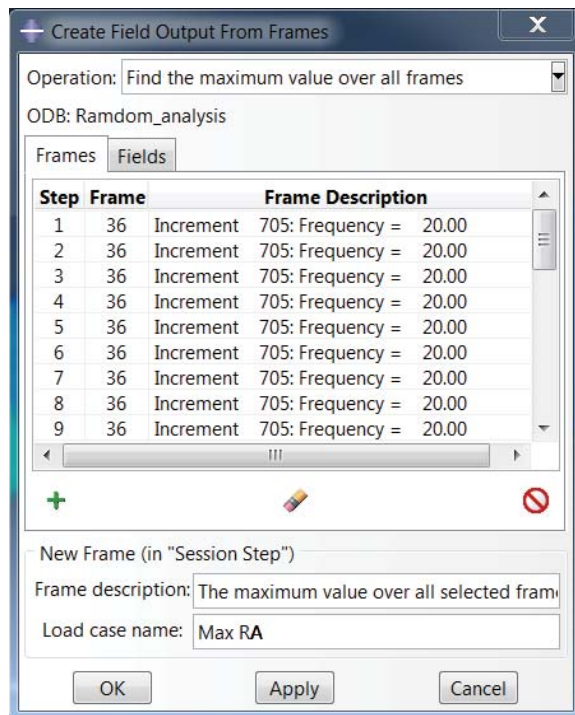
Fig. 10.27   Defining the name of the frame.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
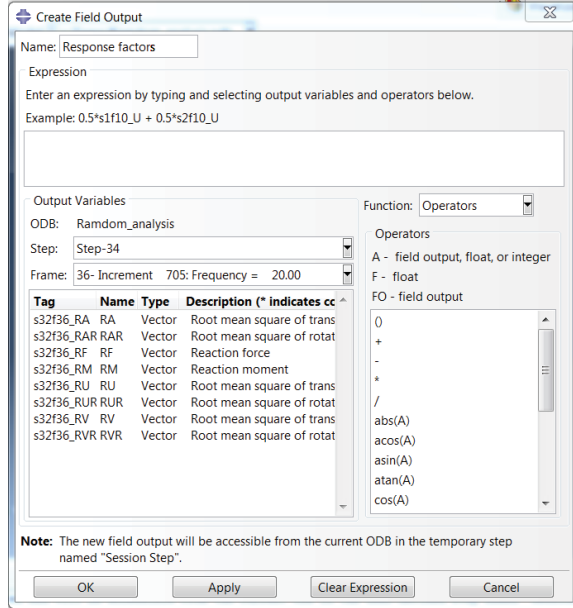
Fig. 10.28   Creating the response factor field output.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
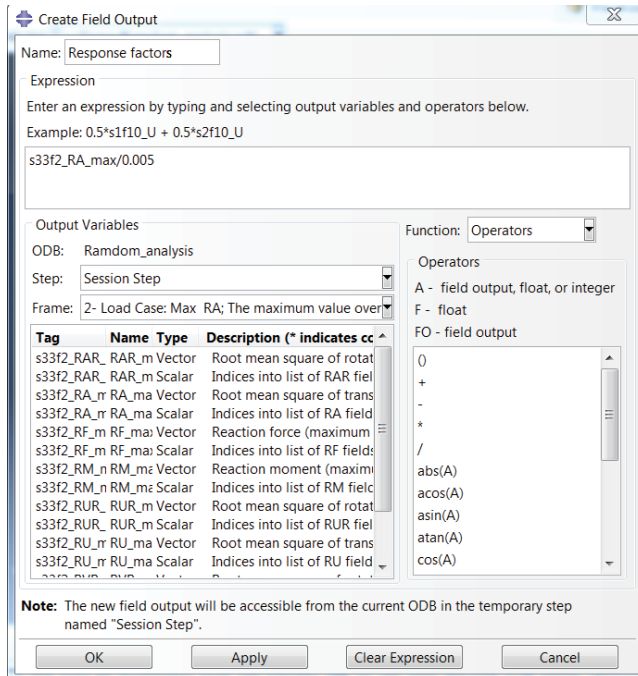
Fig. 10.29   Choosing the session step.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
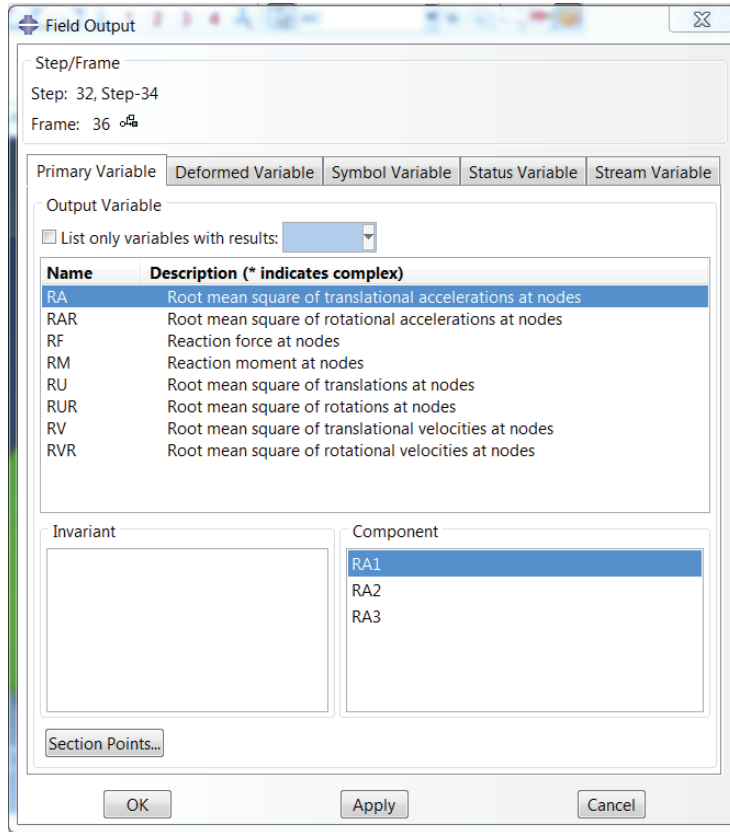
Fig. 10.30   Field output window.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

Step 3: plot response factor map

- Go to *Result – Field Output*.
  A new window will pop up as shown in Figure 10.30.
- Click on *Frame – Field Output*.
- In Figure 10.31, choose *Session Step, Frame 5 the maximum value over all selected frames*, click *OK*.
  A new window will pop up as shown in Figure 10.32.
- Then choose *RA_max*, and choose *RA3* component, which indicates the vertical direction, click *OK*.
- A response factor map is plotted as shown in Figure 10.33.

An engineer can design the slab according to the response factor map plotted above; if the response factor is greater than the required value of the code, remedial measures such as the increasing the slab thickness can be undertaken.
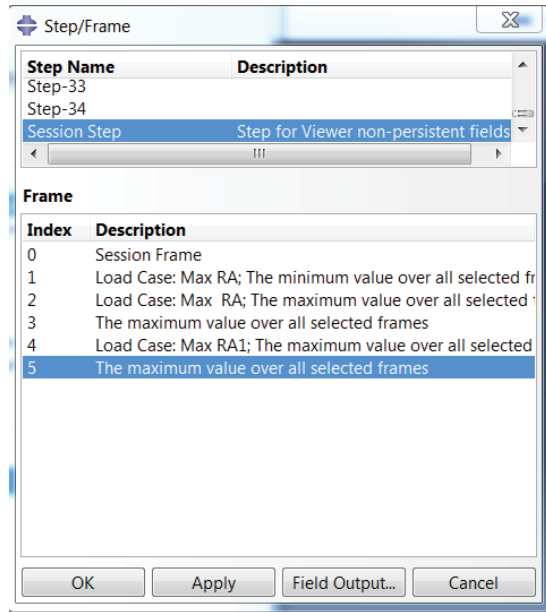
Fig. 10.31   Choosing frame 5.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
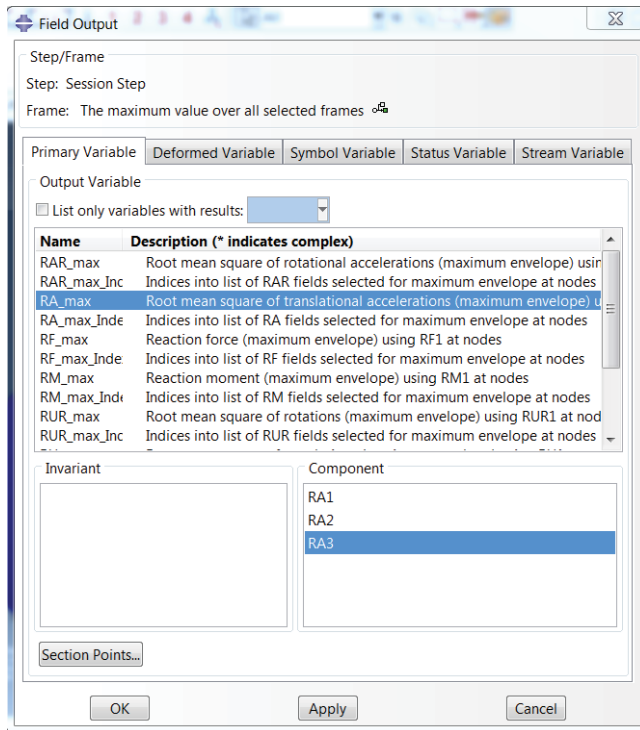
Fig. 10.32   Choosing the vertical root mean square of acceleration.
Abaqus® screenshot reprinted with permission from Dassault Systèmes.
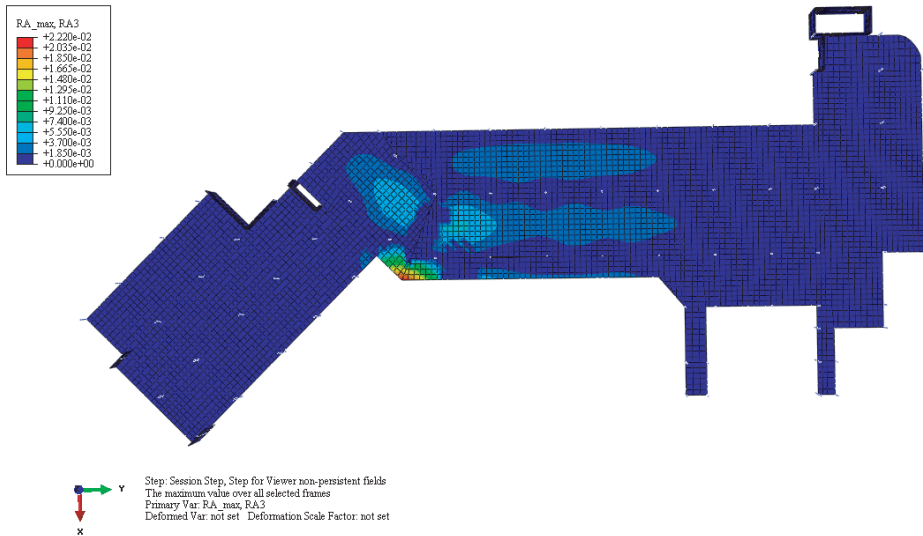
Fig. 10.33 **Contour of the vertical root mean square of acceleration (response factor map).**
Abaqus® screenshot reprinted with permission from Dassault Systèmes.

References

Bachmann, H.L., 2002. "Lively footbridges – a real challenge", Proceedings of the International Conference on the Design and Dynamic Behaviour of Footbridges. Paris.

Brownjohn, J.M.W., Pavic, A. and Omenzetter, P. 2004. "A spectral density approach for modelling continuous vertical forces on pedestrian structures due to walking", *Canadian Journal of Civil Engineering*, 31, 1: 65–77.

Fu, F. 2009. "Progressive collapse analysis of high-rise building with 3-D finite element modelling method", *Journal of Constructional Steel Research*, 65: 1269–1278.

Matsumoto, Y., Sato, S., Nishioka, T., Shiojiri, H. 1972. "A study on design of pedestrian over-bridges." *Transactions of JSCE*, 4: 50–51.

Ove Arup & Partners. 2004. *Hospital Floor Vibration Study – Comparison of Possible Hospital Floor Structures with respect to NHS Vibration Criteria*. The Concrete Centre.

Scott, R.H. 1983. "Technical Note 372 – The short-term moment-curvature relationship for reinforced concrete beams", *Proceedings of the Institution of Civil Engineers, Part 2*, 75: 725–734.

Wheeler, J.E. 1982. "Prediction and control of pedestrian-induced vibration in footbridges." *Journal of the Structural Division*, 108, 9: 2045–2065.

Young, P. 2001. "Improved floor vibration prediction methodologies", *Proceedings of Arup Vibration Seminar on Engineering for Structural Vibration: Current Developments in Research and Practice*. IMechE, London.

Codes and building regulations

BS 6472-2:2008. "Guide to evaluation of human exposure to vibration in buildings. Blast-induced vibration" British Standards Institution, 2008.

BS6472-1:2008, 'Guide to evaluation of human exposure to vibration in buildings. Vibration sources other than blasting." British Standards Institution, 2008.

BS EN1990, Eurocode, 2002. *Basis of Structural Design*, Brussels: European Committee for Standardization.

BS ISO 10137: "Basis for design of structures – Serviceability of buildings and walkways against vibrations", International Organisation for Standardisation, 2007

HTM2045 'Acoustics: Design considerations', NHS Estates, HMSO, 1996.

Software and manuals

Abaqus® Theory Manual. 2013. Version 6.13, Hibbitt, Karlsson and Sorensen, Inc. Pawtucket, R.I.

Index

WILEY END USER LICENSE AGREEMENT