ECCS – European Convention for Constructional Steelwork (Ed.)

Design of Steel Plated Structures with Finite Elements

- design of steel bridges and other plated structures is increasingly FEM-based
- leading European steel design experts explain background and procedure
- examples, benchmarks and verifications support designers

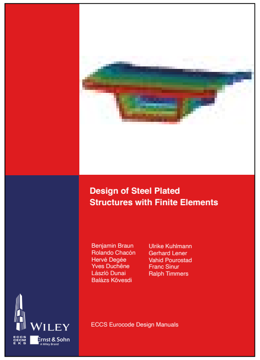
The book deals with the practical design of welded plated steel structures with the finite element method and especially the proof of plate buckling resistance.

ORDER
+49 (0)30 470 31–236
marketing@ernst-und-sohn.de
www.ernst-und-sohn.de/en/3416

* All book prices inclusive VAT.

WILEY

Ernst & Sohn
A Wiley Brand



2023 · 162 pages · 137 figures · 17 tables

Softcover
ISBN 978-3-433-03416-3 **€ 55***

WILEY Ernst & Sohn
A Wiley Brand

ABOUT THE BOOK

Due to the relatively large slenderness of plated structures, the proof of adequate buckling resistance is crucial for dimensioning. Plate buckling is a stability problem of a compression-loaded plate and is generally characterized by failure of the structure prior to achieving plastic component resistance. The stability verification of welded plated structures assisted by finite element analysis (FEM) is becoming increasingly prevalent in applied engineering. The rules and examples given in this book might illustrate the correct application of the numerical models and give background information on the FEM-based design of welded plated structures. Aside from technical background information on calculation methods and software requirements, modelling, solution settings, evaluation, and verification as a key part of this manual, benchmark examples and worked examples are given. They not only allow the user to better understand the procedure and the background but can also serve as test examples to prove the validity of their own numerical calculations. We are sure that this manual will contribute to improved knowledge of the application of FEM for the design of plated structures, and thereby enhance the acceptance of FEM design in practice

ORDER

Quantity	ISBN / Order-No.	Title	Price €
	978-3-433-03416-3	Design of Steel Plated Structures with Finite Elements	€ 55*

☐ Private ☐ Business

Please send your order to:
marketing@ernst-und-sohn.de

Company, Abteilung
VAT-ID
Name
Phone
Fax
Street, No.
Zip code / City / Country
E-Mail

www.ernst-und-sohn.de/en/3416

Date, Signature

. As of 2/2023

TABLE OF CONTENTS

Preface

In steel constructions, slender plated structures which are subject to in-plane forces and are composed of flat, unstiffened or stiffened plates, are designed according to EN 1993-1-5 [1].

Due to the common slenderness of plated structures, the proof of adequate buckling resistance is crucial for dimensioning. Plate buckling is a stability problem of a compression loaded plate and is generally characterised by failure of the structure prior to achieving plastic component resistance.

According to EN 1993-1-5, these buckling verifications may be carried out using two analytical verification concepts; the effective width method (EWM) and the reduced stress method (RSM). A verification assisted by finite element analysis (FEM) is equally permitted.

The application of FEM is becoming increasingly prevalent in applied engineering, despite the fact that this method is currently only briefly dealt with in the informative Annex C of EN 1993-1-5. The reasons for the increasing use of FEM for buckling verification in practice are:

The two analytical buckling verifications are associated with a high calculation effort.

Software manufacturers have prepared their FEM products in such a user-friendly manner that even users with a low level of subject matter expertise can operate the software after a very short period of training / familiarization.

The computers in engineering offices have such a high level of performance that buckling verification can be carried out using FEM within acceptable computing times.

For the 2nd generation of Eurocodes the existing rules concerning FEM, which are spread in different parts of Eurocode 3, will be collected and enhanced to a new individual part prEN 1993-1-14 [2] on “Design assisted by FEM”, which will define the framework of FE design methods for all types of steel structures. As this is still under development the manual given here will mainly refer to the existing rules in EN 1993-1-5 [1]. However, as |prEN 1993-1-14 [2] is partly derived from the content given in Annex C of EN1993-1-5, no contradiction should occur when using the recommendations given here also in future. In contrast, the rules and examples given here might illustrate the future prEN1993-1-14 and its application.

In addition to the consideration of material and geometric non-linearities, the approach of geometrical and structural imperfections plays an essential role for plate buckling design. If the separate consideration of the aforementioned imperfections is not possible, the Euro-code offers the possibility of establishing equivalent geometric imperfection shapes. However, detailed user instructions about how to combine the local and the global, as well as the leading and the accompanying imperfections are not yet provided clearly. As a result, a variety of scenarios with different imperfection patterns have to be taken into account and the associated effort increases considerably.

This is one of the issues where useful information is given within this manual. Aside of technical background information on calculation methods and software requirements, modelling, solution settings, evaluation and verification as a key part of this manual benchmark examples and worked examples are given. They do not only allow the user to better understand the procedure and the background, but can also serve as test examples to prove the validity of own numerical calculations.

We are sure that this manual will contribute to an improved knowledge on the application of FEM for the design of plated structures and thereby enhance the acceptance of FEM design in practice.

Gerhard Lener and Ulrike Kuhlmann, in January 2021

Acknowledgements

Already in May 2008 within the meeting of the Technical Working Group 8.3 (Plates), a sub-group of the ECCS Technical Committee 8 ”Stability”, under the chairmanship of Ulrike Kuhlmann decided to develop an ECCS Manual on ”Design with Finite Elements” focusing on benchmark and worked examples. From that time the project was driven forward mainly also by the former Technical Secretary Benjamin Braun, sometimes with good progress, sometimes slowing down because of various urgent code developments which caught the attention of the members. Since 2010, there has been a close cooperation between TWG 8.3 and the relevant group responsible for the code development of EN 1993-1-5 in the frame of TC 250/SC3, first as Evolution Group later as TC 250/SC3/Working Group 5, including combined meetings. When in 2015 Benjamin Braun resigned from his position as Technical Secretary due to time constraints, Gerhard Lener took more and more over the full responsibility of organizing and finalizing the manual. We are very thankful to him that he led this task up to the very end, when in 2020 a small expert group of TWG 8.3 did a final quality check of the entire document. The following experts (named in alphabetical order) contributed as authors to the manual. The chapters of main contributions are given in brackets, though this is not meant exclusively. In fact, it has been a real cooperative work of the contributors.

Authors:
Braun, Benjamin, Berlin (4.1, 4.2)
Chacon, Rolando, Barcelona (7.4)
Degée, Hervé, Hasselt (5, 6, 7.2, 8.2, 8.3)
Duchéne, Yves, Liége (8.2, 8.3)
Dunai, László, Budapest (4.3, 4.4, 7.3)
Kövesdi, Balázs, Budapest (4.3, 4.4, 5, 6, 7.3)
Lener, Gerhard, Innsbruck
Pourostad, Vahid, Stuttgart (7.1)
Sinur, Frank, Ljubljana (4.4)
Timmers, Ralph, Innsbruck (8.1)

I also want to recognise the continued support and involvement of the various members of TWG8.3, who at the moment are Balázs Kövesdi; Benjamin Braun; Bettina Brune; Chris Hendy; Dieter Ungermann; Dinar Reis Zamith Camotim; Emanuele Maiorana; Enrique Mirambell; Gerhard Lener; Hartmut Pasternak; Heikki Holopainen; Hervé Degée; Ivan Baláž; João Pedro Martins; Joël Raoul; László Dunai; Laurence Davaine; Leroy Gardner; Liv Eltvik; Luís Simões da Silva; Marek Piekarczyk; Marian Gizejowski; Markus Feldmann; Martin Mensinger; Milan Veljkovic; Joseph Ndogmo; Nicole Schillo; Pierre-Olivier Martin; Primož Može; Ralph Timmers; Rolando Chacon and Vincent de Ville.

Finally, we should not forget that two of our former members have passed away, who greatly supported the work either directly or by the spirit of collaboration they contributed to the group, which are Darko Beg, Ljubljana and Bernt Johansson, Lulea.

My sincere thanks to all of them for this excellent work.

Stuttgart, 12/01/2021

Ulrike Kuhlmann

Chapter 4

Modelling (Preprocessing)

4.1 MATERIAL PROPERTIES

The correct representation of material behaviour is of utmost importance because it is one of the most influential parameters that may affect the results of numerical analyses. Generally, mechanical properties of steel are characterised by elastic modulus, yield strength, ultimate strength and strain hardening behaviour. It is common to assume isotropic material with identical values of mechanical properties in all directions so that the input can be based on uniaxial stress-strain data.

Recommendations for material models are given in Annex C.6, EN 1993-1-5 [1], and in Swedish Steel Standard BSK 07 [9]. Basically, EN 1993-1-5 provides three material models, see Figure 4.1. Poisson's ratio for steel is typically set to $\nu = 0.3$ and the elastic modulus is $E = 210\,000$ N/mm^2.

The bilinear stress-strain curves are intended for calculations for which no stress-strain data are explicitely available. If strain hardening is neglected a yield plateau is theoretically assumed, see Figure 4.1a. For reasons of numerical convergence, it might be useful to define a nominal plateau slope with a small value of $E/10000$.

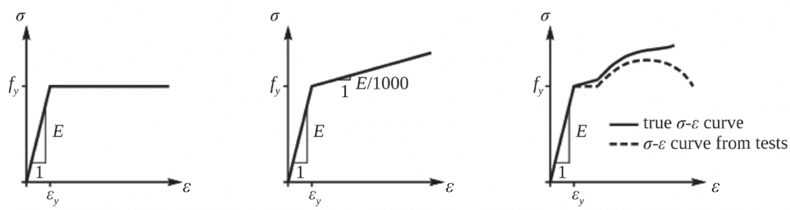
Whether strain hardening is considered or not depends on the required accuracy. Following the comments in [10], besides primary membrane stresses, secondary bending stresses occur in both directions in a buckled plate. Using a yield plateau results in an earlier loss of plate bending stiffness as soon as yielding of the primary stresses starts. Thus, plate buckling will occur also slightly earlier which can be avoided when strain hardening is considered. In case it is considered, EN 1993-1-5 proposes a simple approximation with a slope of

$E/100$ which has proven to be a reasonable value for a wide range of steel grades for strains up to 5 %, see Figure 4.1b.

If material properties from tests are available, the stress-strain curve can be followed one-to-one. Then the stress-strain curve from tests should be transformed into the true stress-strain curve to account for the decrease of cross sectional area near fracture, see Figure 4.1c. The true stress-strain curve can be derived from the stress-strain data of the tests with Eq. (4.1). However, this effect is noticeable only when the strains at failure become very large.

$$\begin{aligned} \sigma_{true} &= \sigma \cdot (1+\varepsilon) \\ \varepsilon_{true} &= \ln(1+\varepsilon) \end{aligned} \tag{4.1}$$

Besides that, BSK 07 offers a quite useful parametrised stress-strain curve according to Figure 4.2. Since it is based on the values from elastic modulus, yield strength and ultimate strength, it allows a very simple though refined definition of the material behaviour.

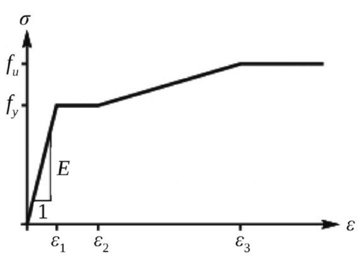


(a) bilinear stress-strain curve without strain hardening

(b) bilinear stress-strain curve with strain hardening

(c) true stress-strain curve based on tests

Figure 4.1 – Material models according to Annex C.6, EN 1993-1-5 [1]



$$\varepsilon_1 = \frac{f_y}{E}$$

$$\varepsilon_2 = 0.025 - 5 \cdot \frac{f_y}{E}$$

$$\varepsilon_3 = 0.02 + 50 \cdot \frac{f_u - f_y}{E}$$

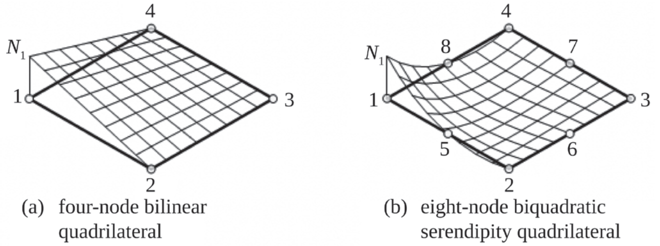
Figure 4.2 – Material models according to BSK 07 [9]

4.2 MESH DISCRETISATION

The description of mesh discretisation hereafter is based on the most common use of shell finite elements. When using them, the geometric mid-surface of the mathematical model is discretised by dividing it into a mesh of finite elements of the chosen type. There are a few shell elements with different features, ranging from elastic shell theory to nonlinear theory with large rotations and plasticity. Nonlinear buckling analysis of thin plates is based upon small strain, large displacement analyses accounting for large rotations. Besides material nonlinearity, the geometrical nonlinear effect in the analysis of slender thin plates is caused by finite (large) rotations. The requirements imposed by the nonlinear analysis lead to the preselection of a shell finite element which accounts for nonlinear, large rotation, and even finite strain applications, plasticity and transverse shear deformation. These elements can be also used for linear bifurcation analysis, e.g. to generate eigenmode-affine imperfection shapes. Because they usually perform similar well as purely elastic shell types, there is no need to change the shell type.

Generally, shell elements have six degrees of freedom (DOF) at each node: translations in the x-, y-, and z-axis, and rotations about the x-, y-, and z-axis. Element types may differ in the type of shape function which is either a bilinear or a biquadratic shape function, see Figure 4.3. The eight-node quadrilateral is often a serendipity element, i.e. without an interior node.

The number of integration points for shell elements is usually five over the thickness. An increase in the number of integration points in regions where high bending stresses over the thickness are expected may overcome possible numerical difficulties.



(a) four-node bilinear quadrilateral

(b) eight-node biquadratic serendipity quadrilateral

Figure 4.3 – Shape functions of quadrilateral shell finite elements (showing a perspective view of the shape function N1)

The piecewise approximation of the mathematical model by finite elements inevitably leads to a discretisation error which should be reduced to a minimum. The accuracy of results may be increased according to [11] as categorised below:

- *h*-method: Overall refinement of the mesh density.
- *r*-method: Refinement of the mesh density in areas with high stress gradients.
- *p*-method: Increase of the polynomial degree of the element's shape function.

It is of utmost importance that the mesh density should be able to cover buckling shapes and stress gradients adequately. Figure 4.4 shows the results of a discretisation study on a square plate under bending for which finite elements with bilinear and biquadratic shape functions have been used. The deviation from the code rules is drawn over the number of elements per edge length. It can be shown that a refinement of the mesh density clearly leads to a reduction of the discretisation error. Besides that, it is found that biquadratic elements show a lower divergence and converge faster than bilinear elements.

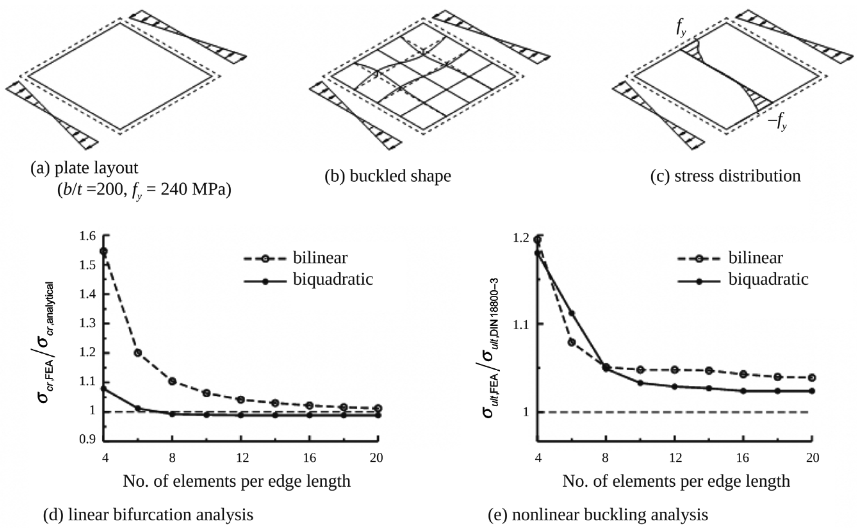


Figure 4.4 – Discretisation study

According to Figure 4.4d, firstly a linear elastic bifurcation analysis was carried out and the first eigenvalue was determined. The quadratic elements converge quickly whereas the linear elements require roughly twice the number of elements. Though it is difficult to give general recommendations, it can be concluded that a sufficient number to cover a buckling halfwave is a number of six elements for the linear shape function and a number of four elements in case the quadratic shape function is used.

Secondly, a nonlinear buckling analysis including imperfections was carried out. At ultimate state, a nonlinear stress distribution occurs which may require a finer mesh density in order to cover the stress gradient adequately. From Figure 4.4e it can be shown that the quadratic element converges rapidly. In contrast to this, the linear element is not too bad, but it is slightly stiffer throughout which leads to slightly higher resistances.

4.3 BOUNDARY CONDITIONS

4.3.1 General

Modelling of the boundary conditions is decisive to ensure the correct structural behaviour of the analysed plate element or structure. The boundary conditions consist of all the support and loading conditions, which should be handled separately in the modelling phase, because their definition needs different considerations and they have different roles in the calculation process.

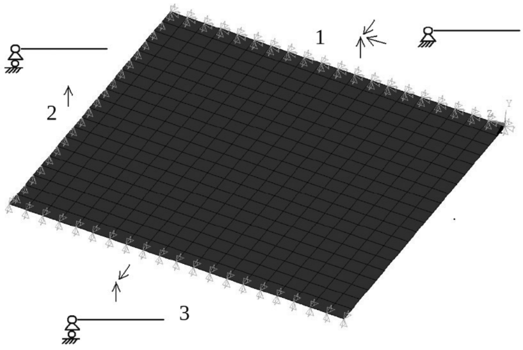
The support conditions in the numerical model should be chosen to reflect in a realistic or conservative manner the behaviour of the physical supports in the real structure. Supports can be defined where direct supports (founding) or supporting structural members exist. The effect of the connecting structural elements can be included in the numerical model without the direct modelling of them by using support conditions, which helps to reduce the FE model size. Supports can be applied as well, if only a part of the entire structure is modelled (sub-models). In this case the stiffness conditions of the environment should be considered by the modelling the supports.

Modelling of loads can be also different. Direct loads or internal forces can be modelled by applying forces and bending moments directly on the nodes in the FE model. In this case the calculation process will be a load-controlled analysis. Loads can be modelled by displacements as well, where the calculation process will be a displacement-controlled analysis.

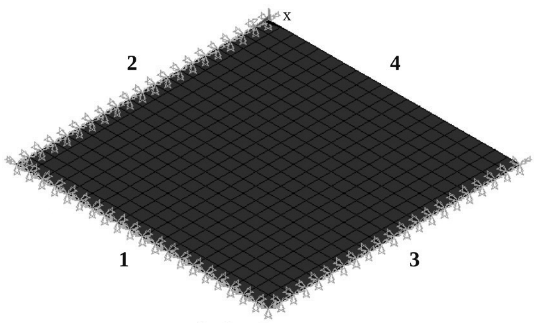
In the following sections possible support and loading models are introduced, which can be applied by modelling of plated structures, or plated structural elements.

4.3.2 Definition of supports

Supports are used to restraint structures against rigid body motions and to represent the supporting effect of the connecting structures or foundations. The applied supports should be in accordance with the used finite element type. Degrees of freedom numbers which are available for the applied finite elements should be considered and the applied support should be harmonized by the DOFs. For the investigation of plated structures shell elements are commonly used having 6 DOFs per node. Volume elements have usually 3 DOFs per node, which has also influence on the applied support conditions. The supports of a simple plate can be modelled along the edges of the plate. The support conditions along each supported edge can be simply supported, semi-rigid or fixed. Fixed and simple supports can be directly applied on the nodes of the plate edges (Figure 4.5). If semi-rigid connections are to be modelled the nodes of the plate should be supported by springs which are connected to supported nodes (Figure 4.6). All the three different support models are presented in Figure 4.5 and Figure 4.6.



a. simple supports



b. fixed supports

Figure 4.5 – Supports for plates

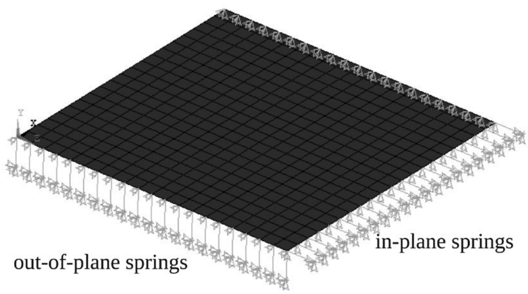


Figure 4.6 – Semi-rigid support conditions for plates

Two type of support conditions can be defined:

- supports against displacements (x, y, z directions),
- supports against rotations (around x, y, z axis).

Both types of supports can be defined directly on the nodes of the FE model. However, they are not independent from each other in case of plated structures. It is possible to define rotational restraints by using only supports against displacements. Application of the displacement restraints along one edge gives a fixed (moment resistant) support to the plate in the perpendicular directions. It means that one must take care about the possible rotational restraints by applying only displacement supports in the model, which can result of unintended rotational support conditions. In the case of most of the engineering structures, this support model (support along the edges) is adequate and fits the boundary conditions of the real structures. But in specific cases the displacement supports along the edges may not give adequate result. There is another possibility to implement a moment restraint in the model presented in Figure 4.7. In this case the vertical displacement of a band of nodes is fixed. The width of the band should be determined according to the stiffness of the plate.

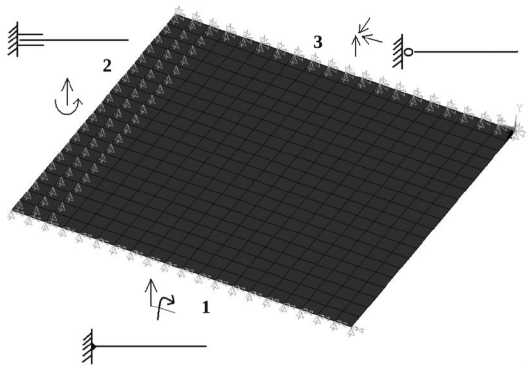


Figure 4.7 – Modelling of moment restrained edges

If stiffness conditions of the connecting members or the supporting members should be implemented in the FE model, semi-rigid support conditions can be applied instead of the rigid supports. The modelling of the semi-rigid supports can be defined by using springs or contact elements. If semi-rigid connection is to be defined, one row of nodes should be determined beside of the plate and the springs or the contact elements should be connected between the node of the plate and the additionally defined nodes. This support method is shown in Figure 4.8. In several FE programs the direction of the offset of the additional row should be in the working direction of the springs. In several FE programs the direction of the offset does not have any influence, the working direction of the springs can be defined by the properties of the element type. In several FE programs, the designer should work with coincident nodes, since some spring elements do not account for finite length, thus giving a moment imbalance. The type of the semi-rigid connection can be different (springs against displacement, springs against rotation, linear springs or non-linear springs), depending on the applied element properties. Using semi-rigid support model, the additionally defined nodes should be also supported avoiding an unconstrained model and rigid body motion.

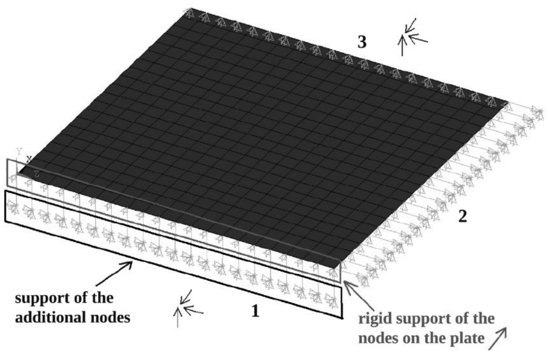


Figure 4.8 – Semi-rigid support model

If the analysed plate is a built-up section (for example with longitudinal stiffeners), the definition of the support conditions should be defined more carefully. If the same support model is used as by the simple plate (support along the plate edges) an eccentricity can be given to the structure, which can have influence on the failure mode and structural behaviour. Especially, in case of buckling analysis, the buckling direction can be significantly influenced by the eccentric supports, therefore special care should be given to the support eccentricity.

A more advanced and recommended solution to model supports on stiffened plates is to apply supports in the center of gravity of the end cross sections. In this case, an extra node should be defined in the cross section center of gravity and all the nodes in the cross section should be coupled to this node by rigid elements or by constrained equations, as shown in Figure 4.9. Constraint equations provide many useful features, such as tying together dissimilar meshes, representing parts of the system not explicitly modelled, or distributing loads and supports. These rigid members or constrained equations ensure that the end cross sections are kept as a plane and by applying pinned or fixed supports in the middle point supports can be modelled. This support model by using the above described rigid end cross section method has many advantages. Beside that pure pinned or fixed connections are possible to be achieved, the rigid end cross section works as a diaphragm as well. It means that the possible local failure at the load introduction location can be avoided, that is usually not preferred and it can reduce the ultimate resistance of the analysed structure. If the effect of the load introduction is investigated and it can have significant effect on the structural behaviour, the use of this support model is not recommended. It has to be noted that the supported nodes should be restrained against rotation along the longitudinal axis to avoid the rotation of the entire structure.

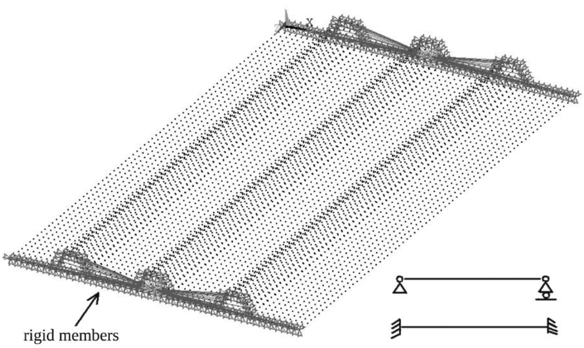


Figure 4.9 – Advanced model of pinned/fixed support model of a stiffened plate

If only displacement restraints are used the end cross section will be fixed against rotation, but the sub-panels of the modelled structure will be still hinged (Figure 4.10a). If the rotations of the sub-panel are also to be fixed, rotational restraints around the axis of the supported edge should be also used at all the nodes in the end cross section (Figure 4.10b). Difference between the two support models can be found by analysing the local buckling of the sub-panels.