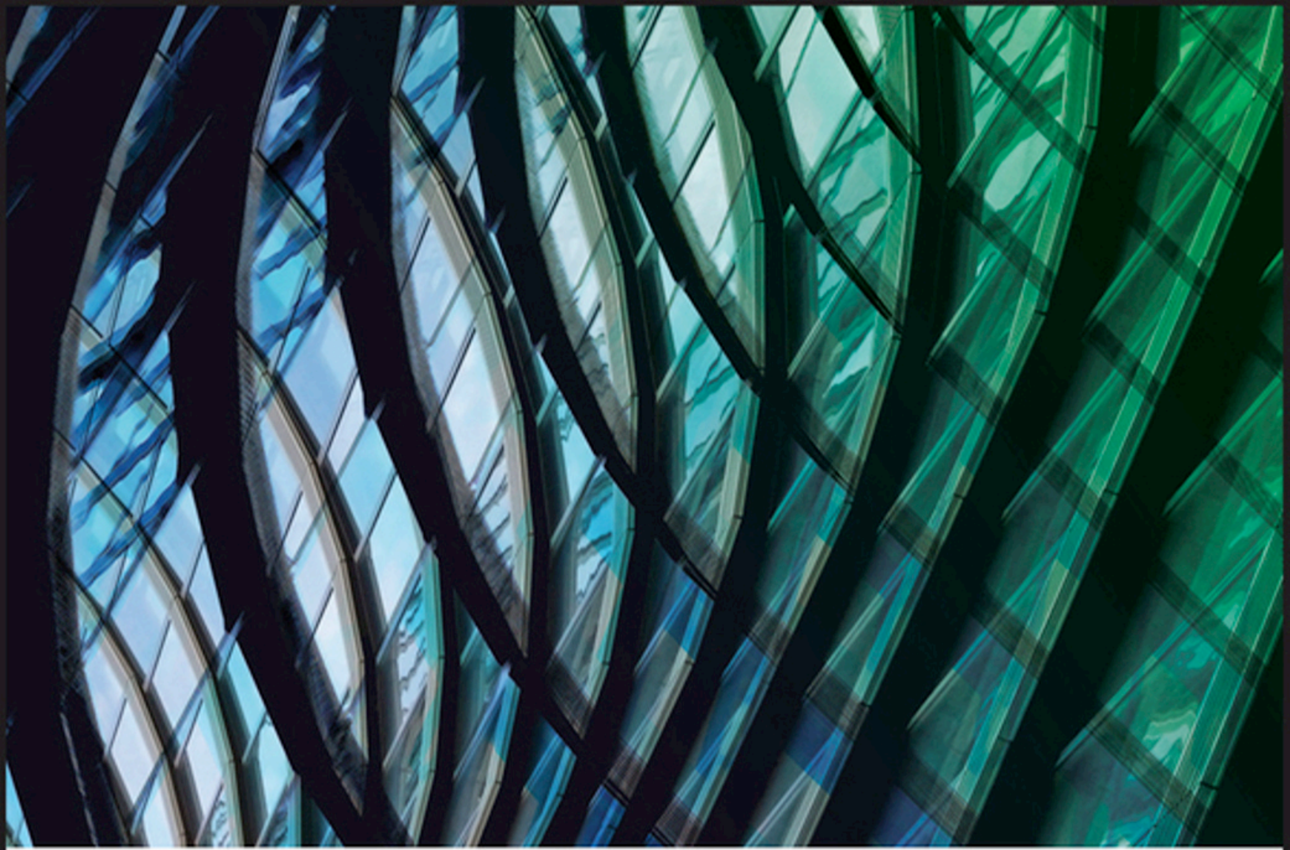

STEEL CONNECTION ANALYSIS

PAOLO RUGARLI

WILEY Blackwell

Steel Connection Analysis

Steel Connection Analysis

Paolo Rugarli
Castalia S.r.l.
Milan, Italy

WILEY Blackwell

This edition first published 2018

Registered Offices
John Wiley & Sons, Inc., 111 River Street, Hoboken, NJ 07030, USA
John Wiley & Sons Ltd, The Atrium, Southern Gate, Chichester, West Sussex, PO19 8SQ, UK

Editorial Office
9600 Garsington Road, Oxford, OX4 2DQ, UK

For details of our global editorial offices, customer services, and more information about Wiley products visit us at www.wiley.com.

Wiley also publishes its books in a variety of electronic formats and by print-on-demand. Some content that appears in standard print versions of this book may not be available in other formats.

Library of Congress Cataloging-in-Publication Data

Names: Rugarli, Paolo, 1963– author.
Title: Steel connection analysis / Paolo Rugarli.
Description: Hoboken, NJ : John Wiley & Sons, 2018. | Includes bibliographical references and index. |
Identifiers: LCCN 2017053858 (print) | LCCN 2018000726 (ebook) | ISBN 9781119303480 (pdf) | ISBN 9781119303534 (epub) | ISBN 9781119303466 (cloth)
Subjects: LCSH: Steel, Structural – Testing. | Finite element method. | Building, Iron and steel–Joints.
Classification: LCC TA475 (ebook) | LCC TA475 .R84 2018 (print) | DDC 624.1/821–dc23
LC record available at https://lccn.loc.gov/2017053858

Cover Design: Wiley
Cover Image: © tonymax/iStockphoto

Set in 10/12pt Warnock by SPi Global, Pondicherry, India

10 9 8 7 6 5 4 3 2 1

To all my dears

Fiorenza dentro da la cerchia antica,
  ond'ella toglie ancora e terza e nona,
  si stava in pace, sobria e pudica.
Non avea catenella, non corona,
  non gonne contigiate, non cintura
  che fosse a veder più che la persona.

Dante, Paradiso, XV, 97-102

Contents

Preface

Around 17 years ago, at the end of the 1990s, when I started my research on steel connections with the aim of developing some reliable and general software, able to tackle, hopefully, every connection, I often felt like giving up. The problem was tremendously complex, and the general rules of mechanics difficult to relate to the problem to be faced; there was a huge gap to be bridged.

Initially I thought that only a system able to learn from the analyst could deal with such a complex problem, learning ad hoc rules to be later applied, case by case. However, I was able to move some steps forward, finding what in the second chapter of this book is named the *jnode*, its analytics, and all the related concepts. My first useful result was detecting equal jnodes. Several years were then necessary to develop the tools needed to create the scene, that is, to place the constituents in their proper position, freely placing them interactively in 3D space, in the specific context of steel connection study. The mechanical problem of connection analysis, to be tackled with a general approach, was however still unsolved. I was prepared to develop an expert system able to learn from the user how to recognize specific subproblems to be faced, by simple ad hoc rules. This was the tentative generalization of the methods widely used by engineers, but was not the solution I was searching for.

Adopting the concept of the force packet, and recognizing that the connections could be classified as isoconnected or hyperconnected, I finally understood, in 2008, that a simplified finite element model that in this work is named IRFEM, could be used to compute the force packets flowing into the connectors for a generic set of connections. Then, by the action and reaction principle, a cornerstone for connection analysis, the forces loading the constituents could be known, and by finite element models of single constituents using plate–shells, coherent and well rooted Von Mises stress maps could be obtained. This is what I call the hybrid approach and is described in Chapter 9 of this book.

The door was then opened for the automatic creation of finite element models of constituents (2008), and from there, in 2012, to the complete automatic modeling of the whole node, using what I call here the pure fem approach (PFEM). This is seen as a special case of the hybrid approach and is discussed in Chapter 10.

What initially seemed an inextricable tangle could indeed be solved in strict observance of the main principles of mechanics and of plasticity theory.

Several issues are still to be better solved, but a general well rooted method is now available, that can be applied to every connection configuration, from the simplest to the most complex. Indeed, I think this is a useful result, because a part of the method can be implemented with relative ease.

I am well aware that several issues are pending and must still be better tackled. However, after many years of solitary work, I think the time has come to explain what I have researched and to propose my work for the attention of my colleagues.

Anything can be improved, but the structural analysis problem of analyzing steel connections having a generic geometrical configuration, regardless of the number of loading combinations, is now solvable with automatic tools.

Paolo Rugarli
Milan, 17 May 2017

1

Introduction

1.1 An Unsolved Problem

Steel connection analysis and checking is one of the most complex problems in structural engineering, and even though we use very powerful computing tools, it is still generally done using very simplistic approaches.

From the point of view of a typical structural engineer, the problem to solve is to design and check *nodes*,[1] not single connections, i.e. a number of connections between a number of different members – maybe tens or even hundreds of load combinations, inclined member axes, and generic stress states. In a typical 3D structure there may be several tens of such nodes (Figure 1.1), or maybe even hundreds, which may be similar, or may be different from one another; identifying nodes that are equal is one of the problems that the designer has to face in order to reduce the number of different possible solutions, and in order to get a rational design. However, this problem of detecting equal nodes has not been sufficiently researched, and there are currently no tools that are able to properly solve this issue.

If posed with the due generality, the problem of checking 3D nodes of real structures has not been solved by automatic computing tools. Also, because a general method of tackling all these problems is apparently still lacking, usually a few "cooking recipes" have been used to solve a limited number of typical, recurring (2D assimilated) nodes. Indeed, it often happens that true, real world nodes have to be analyzed by such recipes, despite the fact that the basic hypotheses needed to apply these recipes do not always hold true. This poses a serious problem because although these "cooking recipes" have been widely used, in the past few years they have been applied to 3D structures designed using computer tools, in the non-linear range, perhaps in seismic areas, and with the aim of reducing the weight of steel.

The effects of such oversimplification have already been seen in many structures where steel connections have failed, especially in seismic areas (e.g. Booth 2014), but even in non-seismic areas (e.g. White et al. 2013, Bruneau et al. 2011). Generally speaking, it is well known that connections are one of the most likely points of weakness of steel structures, one of the most cumbersome to design – indeed one of the least designed – and one of the least software-covered in structural engineering.

1 It will be seen that the term *node* is too generic for the aims of steel connection analysis. In this introductory chapter, however, it will be used due to its widespread diffusion.

Steel Connection Analysis, First Edition. Paolo Rugarli.

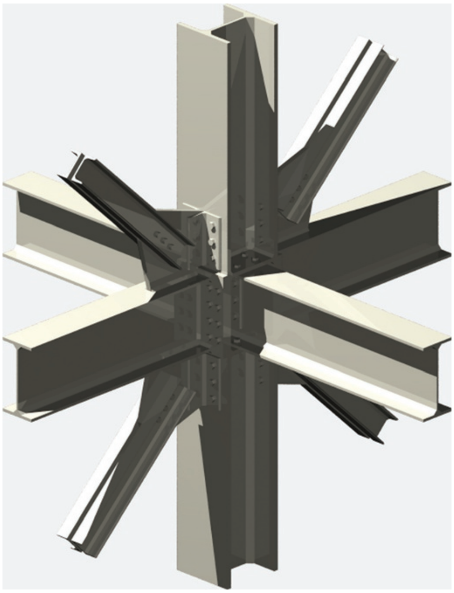

Figure 1.1 A possible *node* of a 3D structure.

This book describes the research efforts made by the author since 1999 to tackle these issues, and it proposes a general set of methods to deal with these problems (see Section 1.6 for more details).

1.2 Limits of Traditional Approaches

1.2.1 Generality

Traditional approaches to connection design have been extensively used for many years, and are still widely used. Usually they imply several simplifying hypotheses, which are needed in order to apply them in by hand computation. The equivalent of by hand computation is today a "simple spreadsheet" often written very quickly for each given job. As with every other form of calculus, they are prone to serious errors (*slips* and *lapses* – see Reason 1990 for a general study of human error, and Rugarli 2014 for a discussion on validation of structural models; for spreadsheets programming errors, see the European Spreadsheet Risk Interest Group web site).

There are several possible design situations where the use of traditional approaches is completely justified. These approaches are rooted in the traditional 1D or 2D design. The use of 2D design needed intense by hand computation or the use of graphic tools up to the 1970s; at that time there was no need and no specific legal requirements for checking tens or may be hundreds load combinations, and safety factors were much higher than those used nowadays. When dealing with such situations, today – for example simple determined structures under elementary actions – the use of traditional approaches is still useful. So, it would not be sensible to exclude them completely. Indeed, they will never lose their utility, especially as one of several possible cross-checking tools that can be used to detect possibly unsound designs.

However, in current design practice, we almost always use 3D methods of analysis applied to highly redundant structures, sometimes in the plastic range, automatic computerized checks, with minimum weight often being a must, and safety factors have been reduced to their minimum. (Currently the material safety factor for limit state design is 1.0 in Eurocode 3. The load safety factor for dead loads is lower than that valid for live loads. The maximum loads are applied with a reduction factor ψ to take into account the reduced probability of contemporary occurrence. All these practices were not, as such, in traditional designs, which means that they used higher safety factors.)

In summary, while traditional design of structures was often simple, 2D, and was designed by making extensive use of safe-side envelopes both for loads and for resistance, today things are not so easy; indeed, they are much more complex. While virtually all design steps have been semi-automated (modeling, checking members, drawing them, and even cutting them into true 3D pieces by means of *computer numerical control,* CNC), the checking of connections has remained at the traditional level, more or less upgraded to the modern era by the use of spreadsheets and dedicated, ad hoc software.

As mentioned, several simplifications are widely used in traditional approaches. The following sections will briefly summarize them.

1.2.2 Member Stress State Oversimplification

Members in highly redundant 3D structures are often nonsymmetrical (such as in industrial plants or architects' innovative designs), and under the effect of combined load cases, they are always loaded in the most general way. If they are not: (a) fully hinged at both extremities, (b) straight, and (c) with no transverse load applied, they will in general exhibit all six internal forces components: an axial force, two shears, one torque and two bending moments, referred to the principal axes of the member cross-section.

Idealizing the connection in such a way that some member internal forces components are considered zero at the connection is still a widely used practice. While this is justified when the connections are specifically conceived with that aim, this is unjustified for connections that are not so designed. In a typical moment resisting frame (MRF) ideally designed to work in a plane, beam-to-column connections that must transfer bending and shear in one plane (and axial force) will always transfer the bending and the shear *also in the other plane* – and of course torque. So the internal forces to deal with are not three, but six. Sometimes it is said that the torque and out-of-plane bending are avoided by "the concrete slab", or by something equivalent, but often the concrete slab does not exist or cannot be considered a true restraint, or its true effect is questionable.

A simple beam hinged at an extremity (e.g. Figure 1.2), will transfer the shear, and will not transfer bending moment if the connection is light and does not use flanges, but it will also transfer the axial force and, if any, the shear in the other direction. However, textbooks usually refer to "shear connections" and only recently, under the flag of "robust design" (a replacement for *correct design*) has this axial force finally – sometimes, in some textbooks, – been considered (e.g. the Green Books by SCI).

This systematic neglect of some internal forces which have, however, been computed introduces a clear mismatch in the design process. Simply, load paths are interrupted (Figure 1.3) and the corresponding forces are thrown away: recalling the *variational crimes* of the finite element literature, this can be called a *connection-design crime,* more specifically an *equilibrium crime*. Usually no one cares, and no one mention it.

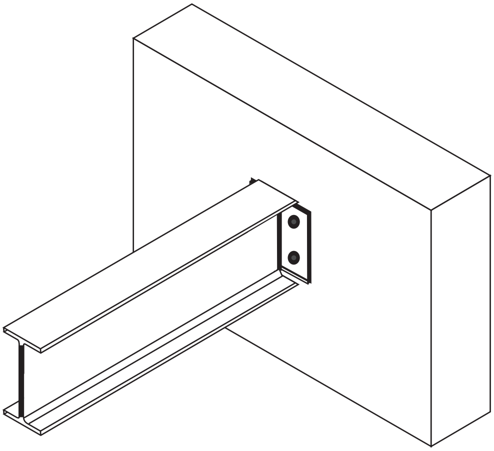

Figure 1.2 Flexible end plate connection ("shear" connection).

Figure 1.3 Traditional design applied to computerized analysis: no way for the load path.

1.2.3 Single Constituent Internal Combined Effects Linearization

Not only are some components of member internal stress states thrown away, but the remaining components are tackled one at a time, as if the connection were loaded only by a single member internal force component. The typical example is the axial force plus (one) bending moment loading condition, for beam-to-column connection or for a base plate. As already pointed out, this loading condition is itself usually the result of a connection-design crime. However, several possible combinations of N and M can be applied to the connection (two infinities), leading to an infinite number of possible stress states. This is usually tackled by computing the limit for the axial force, N_{lim}, and for the bending moment, M_{lim}, as if they were acting alone, and then the mutual interaction is computed by simply drawing a straight line in the (N, M) plane. So the design safety condition becomes

$$U = \frac{N}{N_{\mathrm{lim}}} + \frac{M}{M_{\mathrm{lim}}} \leq 1.0 \tag{1.1}$$

where the utilization ratio, U, can be considered as the reciprocal of the "limit" multiplier $\lambda = 1/U$. It must be underlined that this limit condition is not applied to the member cross-section, but to the member *connections*, implicitly considering all the possible failure modes: bolt bearing, block tear, generic resistance of constituents, buckling of plates, punching shear, weld-resistance, and so on.

As there are quite a number of plastic failures implicitly included in the typical design formulae (e.g. bolt bearing), this must not be considered as a superposition of effects, which would only be valid in linear range.

It must instead be considered a simplification of the limit domain, assuming that it can safely be considered convex, so that a straight line would be a safe simplification; as can be seen, the previous equation is the equation of a straight line in the (N, M) plane.

There are several issues to be discussed here.

The first is that this choice clearly lays aside every possible "realistic" computation of a safety measure. In particular, the *utilization ratio U*, a pure number and a much used safety index, which must be lower than 1 in the safety region, is usually computed as U = PO/AO (Figure 1.4), that is, the ratio of the distance of the applied stress state P (N, M) from the origin, to the distance of point A from the origin, A being the point where the straight line joining P and O meets the limit (linearized) domain. If the true domain were convex the correct utilization ratio would have been U = PO/QO, which can be much lower. So this method is not very realistic, and can be too much on the safe side.

By posing

$$\lambda_N = N_{\mathrm{lim}}/N$$
$$\lambda_M = M_{\mathrm{lim}}/M$$

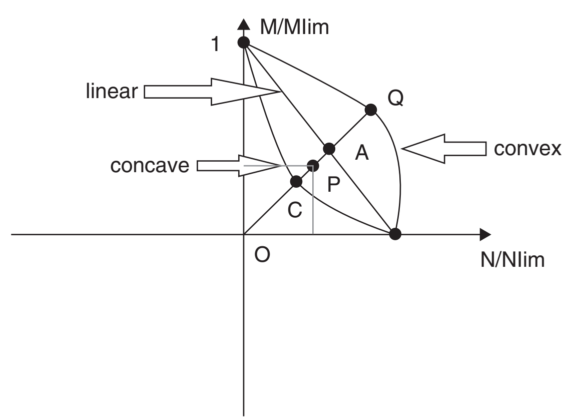

Figure 1.4 Limit domain for a connection; **P**(N, M) is the stress state for a single member assumed.

Equation 1.1. becomes

$$\frac{1}{\lambda_{linear}} = \frac{1}{\lambda_N} + \frac{1}{\lambda_M} \leq 1$$

A similar result can be found in Fraldi et al. 2010, a paper dealing with the problem of finding some bound of the limit multiplier under combined loadings, once the limit multipliers of single loadings are known; there it is formally proved, in the framework of classic plasticity, that the "true" combined-loading multiplier λ, is surely such as to satisfy the following inequality:

$$\lambda \geq \left(\frac{1}{\lambda_N} + \frac{1}{\lambda_M}\right)^{-1} = \lambda_{linear}$$

which, considering Figure 1.4, simply means

$$\mathbf{QO} \geq \mathbf{AO}$$

The second issue is that in order to be confident that the limit domain, considering all the possible failure modes, is convex, no buckling effect must be possible at load levels lower than those leading to the first failure mode, that is, the failure mode which is met first, linearly increasing the stress state from (0, 0) to (N, M). If this is the case, then $U = \mathbf{PO}/\mathbf{CO}$ which is much higher than $\mathbf{PO}/\mathbf{AO}$. A good design of connection should always ensure that the first failure mode is plastic, i.e. ductile, and avoid brittle failure modes. However, this cannot be considered an implicit condition but must be assured by correct sizing and proper numerical checks, which presents a serious problem.

The third issue is related to signs. If the connection is not doubly symmetrical, then it can be expected that reversing signs can lead to different limit values, perhaps due to the buckling effects which must, however, always be kept in consideration, if only to be proven irrelevant. So, to be applicable when signs are reversed, two points in plane (N, M) are not enough and four must instead be evaluated, doubling the effort.

It can be concluded that the practice of drawing a linear domain, considering only the equilibrium-crime survivors, has several limits and can also be: (a) too much on the safe side because the "true" limit multiplier can be much higher than that obtained by the linearized limit domain, (b) *not on the safe side*, if some buckling mode (possibly associated with sign reversal) has not properly been accounted for and checked.

This is not an academic discussion. The point Q which would be obtained by increasing the couple (N, M) and searching for the first failure mode, in classic plasticity, may well be quite different from point A. This can be understood when considering for instance the *plastic* yield lines related to the possible failure modes: the plastic-limit resistance failure of parts of the member itself or of its connected constituents. These yield lines *are tightly related to the load configuration applied*. With only load N (load increasing along the horizontal axis of Figure 1.4), a yield lines set would be found, related to some mechanism. With only load M applied (load along the vertical axis of Figure 1.4), another yield lines set would be found. With a possible combination of N and M, a third, possibly completely different yield lines set would be found (load along the inclined straight line in Figure 1.4). So, linearizing means forgetting the true load state and doing a purely numerical simplification, with no physical meaning.

Finally, premature buckling of connections subconstituents is one of the most frequent failure modes, especially in seismic areas where internal forces sign reversal is a normal condition. So the problem of its correct evaluation is a real problem.

1.2.4 Single-Constituent External Combined-Effects Neglect

One typical simplification of traditional connection design is that when dealing with a node where *n* members are joined, the connections are evaluated one by one, considering two or at most three members at a time, and no more. So, if the node is like A-B-C, where B is the main member and A and C are secondary, connection design is often carried on by considering A-B, and so checking A, B, and their connectors, and then B-C, and so once more B, then C, and then their connectors. However, this working method is not correct, as the effects on B of A and C are contemporary, and not separate.

Consider for instance the node of Figure 1.5: all the five "slave" members act together on the column, which is the "master" member used as a reference for all other members. Considering the effects of each slave member on the column separately would only be possible if the superposition principle were to hold true (and it would not in plastic analysis), and only if the – very different – effects of each member on the column were correctly summed. But this is not what is usually done in designing connections. Usually, the effects will at most be grouped considering typical member configurations, like two opposite beams joined to a web, or two opposite beams joined to flanges, and traditional methods are definitely not able to take all these members into account.

What is clearly dangerous here is the possibility that combined effects could drive the common member to failure, or, more generally, the common constituents.

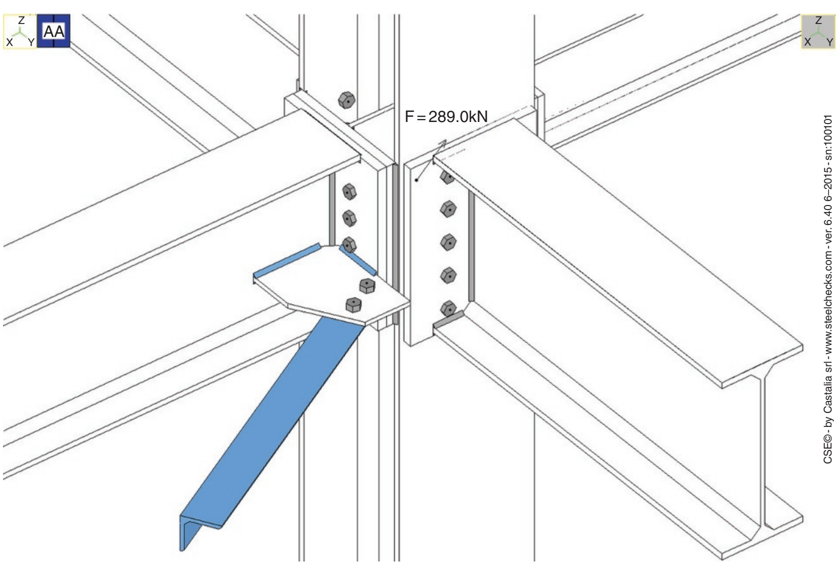

Figure 1.5 All the members connected to the column do act over it at the same time.

Without a clear and coherent computational method to consider the sum of the effects of all the connections, the evaluation of the combined effects is often left to improbable envelopes or the sum of physically meaningless quantities. And this is a very strong weakness.

Besides, it is important to note that the problem is not only related to *master members*, but in general to all constituents. As all the members in a node are directly or indirectly tied together, it is not unusual for the internal forces acting in a member can flow to the connections of another member, a possibility that is implicitly excluded by traditional design methods and that can instead be easily observed when a complete finite element model of a node is set up.

Connections are definitely more complex than a one-to-one, or a one-to-two relationship between members. Connections have inside themselves the same complexity as the whole structure.

1.2.5 Neglecting Eccentricities

True structures and true connections very often have relevant eccentricities that should be ideally considered, but that are very often neglected.

The first type of eccentricity is that of members' axis lines, which in the actual construction are often not as in the finite element model. This leads to possibly severe additional moments that can induce stresses comparable to, or even higher than, those computed considering members to be fully aligned with their computational schemata. As it is very lengthy to properly take into account all these eccentricities by hand, in a 3D context, it is still dangerously considered normal practice to neglect them. This choice is strengthened by the *equilibrium crime*, which, neglecting some internal forces components, also neglects the additional moments that they might drive in some constituents of the connections. However, if a force F is offset by e, the additional moment is Fe, which, assuming a resisting lever b, leads to an additional force equal to Fe/b. If e is much lower than b, then the additional force is negligible, but if it is not, the additional force may considerably increase the nominal one.

To compute the additional moment, in a 3D context, a more precise rule would be (**P** is the true point of application, **O** the point where the force is moved to, **F** is the force applied)

$$\mathbf{M} = (\mathbf{P} - \mathbf{O}) \wedge \mathbf{F}$$

and there are three moment components.

The second type of eccentricity is in the connection area where, due to constructional needs, the true layout of bolts, welds, stiffeners, and cleats may not be that effectively assumed in the connection simplified – often 2D – computational model, especially when considering forces and moments flowing into connectors. Indeed, this computation requires ideally simple, but in true practice quite boring and error-prone vector products. For instance, considering the angle connected to the node in Figure 1.6, while the eccentricity in an horizontal plane will probably be considered, the eccentricity in the vertical plane (the center of the diagonal is not at the elevation of plate mid-thickness) will probably be neglected, which is not correct.

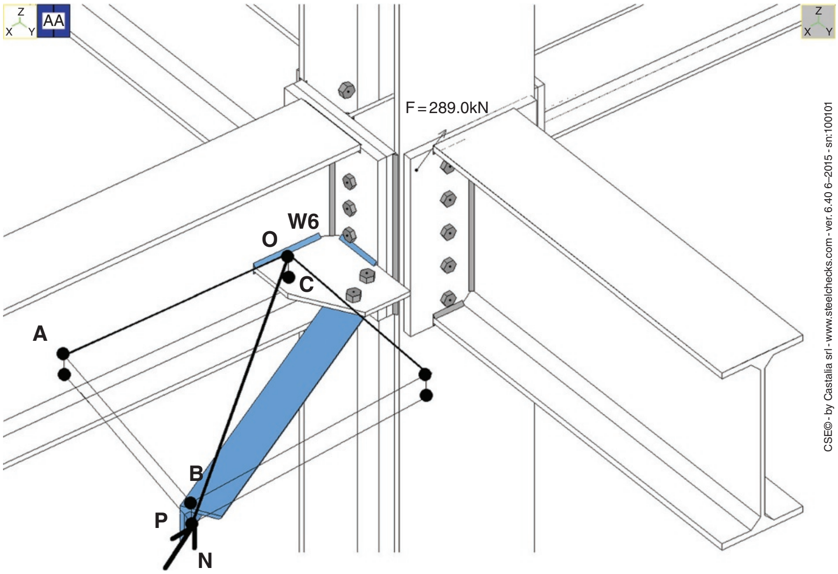

Figure 1.6 Eccentricity between the point of application of the force *N*, P, and the weld layout center, W6, O. The vector (P-O) has the three components (A-O), (B-A) and (P-B), but frequently (P-B) is neglected.

1.2.6 Use of Envelopes

One of the key features of traditional approaches is that they had to deal with a limited number of loading conditions, usually computed by considering envelopes of notional load cases that were themselves envelopes (maximum wind, plus maximum snow, and so on).

Nowadays the number of loading conditions, expressed in the form of load combinations, is quite high. Referring to Eurocodes and applying the combination rules there provided, it is not unusual to get hundreds or even thousands of combinations (Rugarli 2004). This means that the traditional way of computing connections, two or more loading conditions, can be obtained by only assuming a special kind of envelope, which considers maximum or minimum values of different internal forces components acting together. Owing to the equilibrium crime the number of different components of internal forces is often limited to two, so the notional combinations are usually quite a few.[2] Is that approach on the safe side? Well, provided that, when increasing the absolute value of the internal forces all the load effects do always increase, then this might be the case. Unfortunately, this is not always true, and it is not true when a decrease of some internal force, for some failure modes, is more dangerous than an increase, such as in slip checks under compression, or when a specific mix of internal forces leads to worst results.

2 The set of load combinations has to be really complete, otherwise the “maximum” is not really the worst value that the structure might experience, but this is another problem, well rooted in using “realistic” combination sets and not unrealistic, but enveloping.

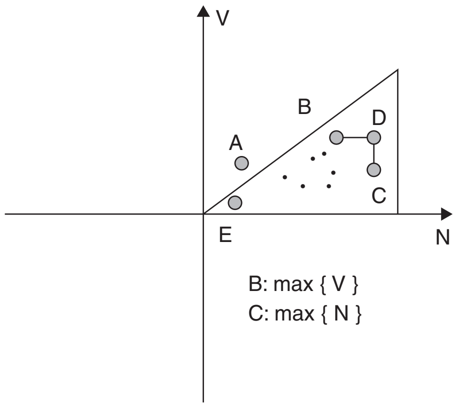

Figure 1.7 Friction connection. Point A is outside the limit domain, point D is inside.

Consider a very simplified example (Figure 1.7), a slip resistant simple support connection relying on the compression N acting over a friction plane having friction constant k. Let N_{lim} be the maximum compression allowable without failure (vertical line bound). A shear V is also applied, and it must be that $V < kN$ to avoid slip. Several couples (N, V) have been computed in a number of load combinations. To check the connection, the maximum N is taken with maximum V. Is that safe? Figure 1.7 clearly shows that it is not. The limit domain is here a triangle.

Point A, neglected by this "envelope" rule, is outside the limit domain. Point D is obtained by mixing maximum N (point C) with maximum V (point B). So considering the maximum absolute values as acting together is not always a safe approach, at least when slip can be a failure mode. It's interesting to note also that adding to checks point E beside point D, i.e. the point with minimum N, the problem would not have been solved, and only mixing the minimum N with the maximum V would have been on the safe side, but possibly, too much.

Indeed, mixing maxima and/or minima can be a quite over-safe approach. Assuming that the governing failure mode has a linear limit domain (Figure 1.8), this way of computing can sometimes really lead to an overestimate of utilization.

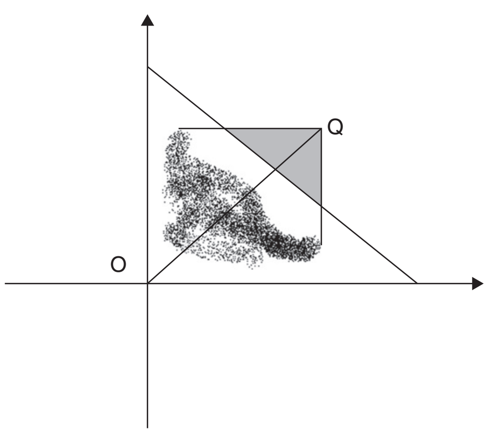

Figure 1.8 Overevaluation of utilization factor using contemporary maxima.

1.2.7 Oversimplification of Plastic Mechanisms Evaluation

Traditional by hand or by spreadsheet approaches all try to evaluate the plastic limit load of complex 3D assembly of steel plates. As this is in general a very complex task, it is not surprising that the number of such computations is reduced to the minimum possible (as it has been shown, by linearizing the limit domain), and that quite often the problem is tackled by using simplified and quite regular geometries.

One problem here is that real world connections do not always comply with such geometries, so the analyst is pushed to force his/her problem into one that is solvable. According to some computational tools, the problem to be solved is always a T-stub, pulled, compressed, or bent, with regular bolt "rows",. But this is not always realistic – for example see Figure 1.9.

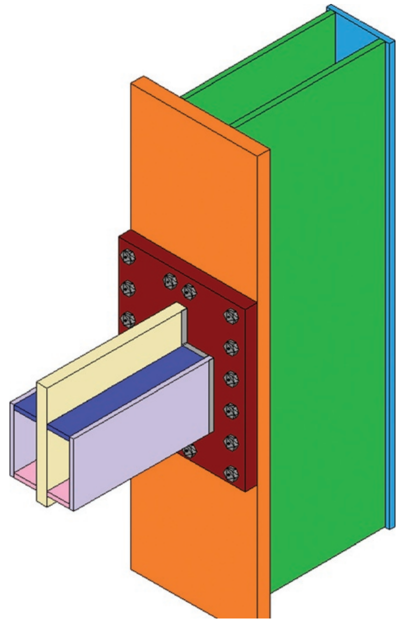

Figure 1.9 Real world moment connection (courtesy CE-N Civil Engineering Network, Bochum, Germany).

Not only can the geometry of the steel plates be quite different, but the bolt layout (not to mention the loading condition see Section 1.2.2), can be different. So, if for some good reason a bolt or a stiffener has to be shifted, or if a plate is not rectangular, or if the footprint of the cross-section and stiffeners is not as regular as in the textbooks, the computational model is simply not able to deal with the problem. Much serious effort has been spent in order to categorize the local failure modes related to typical connections, so that the limit multiplier under simple loads (axial force, bending moment) of typical assemblies could be evaluated. For instance, the *Green Book* referring to moment resisting connections (MRC), published by Steel Construction Institute (SCI 1995), is an excellent book which lists all the possible failure modes and partial yield lines related to typical connections, basically considering the T-stub idealization.

Evaluating the limit load is then a matter of summing up different contributions, analyzing different possible failure lines and modes (Figure 1.10), and finally getting the minimum value. Table 2.4 of the Green Book for moment connections lists 11 patterns for elementary yield lines,[3] used in order to evaluate a final effective length, L_{eff}, to be introduced in the formula for the problem at hand (see also Eurocode 3, Part 1.8, §6.2 and subsections):

$$M = \frac{L_{eff}\, t^2 f_y}{4}$$

3 They are: (i) circular yielding, (ii) side yielding, (iii) side yielding near beam flange or stiffener, (iv) side yielding between two stiffeners, (v) corner yielding, (vi) corner yielding near a stiffener, (vii) double curvature, (viii) group end yielding, (ix) corner yielding, (x) individual end yielding, (xi) circular yielding. For each of these patterns an L_{eff} formula is provided.

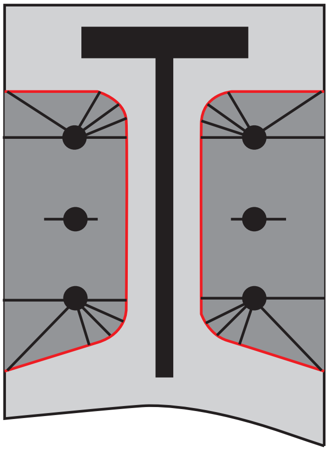

Figure 1.10 An example of evaluation of L_{eff} by summing effects. Red lines are the yield lines whose total length has to be evaluated.

where t is the thickness of the plate where yield lines will appear (e.g. a base plate, an end plate, or a cap plate of a column), f_y is its yield strength, and M is a limit bending moment. The formula is clearly notional and is exact when considering the plastic moment of a plate of length L_{eff} and of thickness t. The use of the typical patterns is not easy, nor particularly intuitive, and it may well lead to errors that are hardly detectable; it is up to the user of such tables to properly mix and compose the typical patterns in a reliable way (see Figure 1.10). Single effects are evaluated and then summed. For instance, to get the L_{eff} of "a bolt row below the beam flange of a flush end plate", we have to evaluate the final L_{eff} related to that bolt row as a function of the individual patterns $L_{eff,i}$ as follows:

$$Min\left\{Max\left\{\left(\frac{ii+iii}{2}\right), ii\right\}, i\right\}$$

but only if some specific geometrical limitations are met, otherwise we have to use

$$Min\{Max\{ii, iii\}, i\}$$

where "i" means "pattern number i". Of course this takes into account only that bolt row.

It is not necessary to get further into that here, but it has to be realized that the evaluation of the limit load multiplier is out of reach of these methods when applied to generic 3D models loading conditions. So, the problem should in general be tackled for what it is: there are six internal forces flowing at the end of each member, many load combinations, many failure modes, a geometry that may well not be forcible into a T-stub, bolt rows that could well be moved (or perhaps no *row* may be available), and so on.

Classical simplified resistance checks continued to use some kind of simplified geometry for the flow-lines of stresses, widely used for instance in the *strut and tie method* (STM), and these are simplified ways of computing plastic mechanisms. For instance, assuming a 30° line of stress flow (e.g. see AISC Steel Construction Manual, 14th ed. §9, "Whitmore section"), and computing some "effective" resistance cross-section to evaluate its limit or ultimate load, is one of these simplified and not always sufficient ways of computing a plastic mechanism. Often, approximations of this type are applied to gusset plates under complex membrane stress (Figure 1.11). Cutting a complex solid with an ideal plane and computing a *net-cross-section* to be checked in a beam-like way is another frequently used simplification. Using simple structural schemata, such as cantilevers, simply supported beams, or struts and ties, extracted from more complex 3D scenes, to be checked for their limit plastic loads is another way to try to assess plastic mechanisms of complex structural configurations. So, when considering what could be called "generic resistance checks", the traditional approaches try to evaluate a plastic mechanism by means of simplified tools. Basically, a generic resistance check is a check that the constituent – usually having an irregular shape – is able to carry the loads applied with no

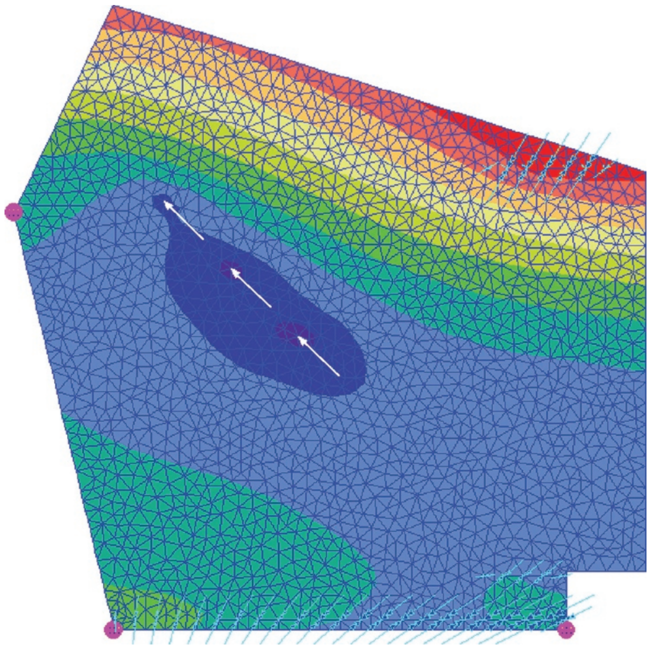

Figure 1.11 Gusset plate under membrane stresses: the three forces in the central area simulate the action transferred by the bolts; weld forces are balanced; fictitious constraints.

plastic mechanism. As local failure modes are tackled by specific checks which can neglect the global configurations (e.g. bolt bearing or punching shear), they are not mentioned here.

1.2.8 Evaluation of Buckling Phenomena

Buckling of some constituent is a dangerous failure mode when it happens before plastic mechanisms have fully developed. Traditional designs cope with this problem in two ways:

1) Size the constituents so that appropriate (low) width-to-thickness ratio of plates is used (e.g. the stiffeners have a thickness at least equal to that of the thickest constituents stiffened).
2) Use simple formulae that model the buckling of complex structural configurations (of geometry and loads) using simple schemata such as the one that is also used to evaluate plastic mechanisms (simple beams, usually "cut from" the existing constituents).

These methods of design work well in many situations, but there are configurations that need a more refined approach. As the load configuration is quite important, a buckling check should be done for every load combination. Often, in traditional approaches, the problem is simply neglected, and no formal proof that buckling modes may not occur is given.

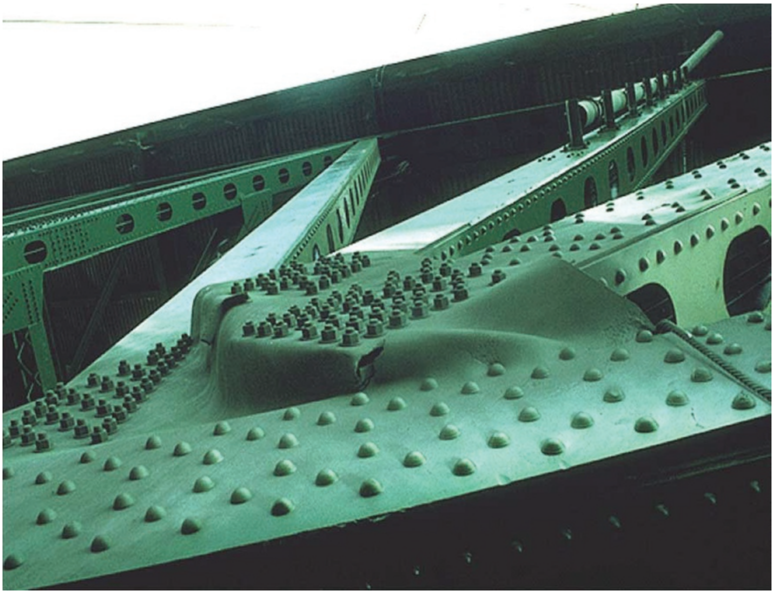

Figure 1.12 Bridge gusset plate buckling failure (from Huckelbridge, A.A., Palmer, D.A. and Snyder, R.E., 1997, "Grand Gusset Failure," Civil Engineering, Vol. 67, September, pp. 50–52. Reprinted with permission of Arthur Huckelbridge).

Neglecting the importance of complex buckling modes related to complex geometries has caused severe problems in many real world structures.

This problem is serious and presently still often unmentioned.

For instance, one of the problems usually tackled by possibly oversimplified approaches is that of the gusset plates. They are under complex force and constraint patterns, and sometimes the methods traditionally used to deal with the problem have shown their limits (see the important photograph taken by Prof. Huckelbridge – Figure 1.12).

1.3 Some Limits of the Codes of Practice

1.3.1 Problem of Coded Standards

Coded standards have different use and meaning depending on the country where they are applied. Sometimes, coded standards are just "advice" with no enforcing value, but other are actual laws of the State and violating them is a legal infringement. In Italy, large books list all the coded rules, and the designer is forced by law to respect them. As well explained in the context of the Three Mile Island nuclear power plant accident, a "big book of rules" is the best way to guarantee a formal respect without actually accomplishing anything. In Kemeny's report on the Three Mile Island accident (Kemeny 1979), it says:

> We note a preoccupation with regulations. It is, of course, the responsibility of the Nuclear Regulatory Commission to issue regulations to assure the safety of

> nuclear power plants. However, we are convinced that regulations alone cannot assure safety. Indeed, once regulations become as voluminous and complex as those regulations now in place, they can serve as a negative factor in nuclear safety. The regulations are so complex that immense efforts are required by the utility, by its suppliers, and by the NRC to assure that regulations are complied with. The satisfaction of regulatory requirements is equated with safety. This Commission believes that it is an absorbing concern with safety that will bring about safety – not just the meeting of narrowly prescribed and complex regulations.

It is the author's opinion that in Italy and also possibly in Europe generally, the boundary line between reasonable and unreasonable regulations has been crossed, especially when considering that in some countries these regulations are enforced by law, and this is true not only for the most general principles, but also for the most detailed ad hoc formulae referring to very special cases. In any case, the hypothetical list of all the needed rules, on the one hand is surely incomplete and possibly even wrong (as these rules frequently assume regular schemata and loads and forget important parts of the problem), and on the other hand this so-specific list is the perfect alibi for people wishing to respect the letter of the rules while violating their deepest meaning. For instance, the Emilia Italian region was declared "not seismic" according to the law. Thousands of precast-concrete industrial buildings using portal frames with no true connection between the beam and the columns were built in the area, and so the moderate 2012 Emilia earthquake led to severe losses. A more discerning approach would perhaps have saved lives.

Referring to steel structure connections, it is not always clearly understood that the specific rules of the coded standards are – by definition, you might say – not complete or even applicable to all situations. For instance, not all connections' plastic mechanisms can be forced into those of a T-stub (described in part 6 of Eurocode 3, Part 1.8, "Structural joints connecting H or I sections), but it is not unknown to see local authorities denying the approval of a design, as the expected T-stubs are not even mentioned in the design reports. Coded standards cannot replace a serious professional judgment, and should never be considered a "by law" alternative to specific design considerations deeply rooted in the best "standards" we have: the laws of mechanics.

In summary, it is the author's opinion that the standards enforced by law should be short and should refer only to general principles. Specific guidance can be written for specific well-delimited problems, while the (humanly understandable) wish to cover all the matters by means of what are in fact specific tools, should be resisted. The way the coded standards are written should clearly push the readers (possessing different levels of knowledge on the matter) to understand that specific problems may well have specific solutions, and that only the general principles of mechanics should always hold true, not the ad hoc specific methods conceived to deal with well delimited problems.

1.3.2 T-Stub in Eurocode 3

One of the most complex failure modes to be investigated is that of "generic resistance", a generic set of constituents not falling into any of the available simplified theories (such as beam theory or rectangular or circular thin plate theory), that is loaded by a complex set of forces and must be checked against plastic limit and other failure modes.

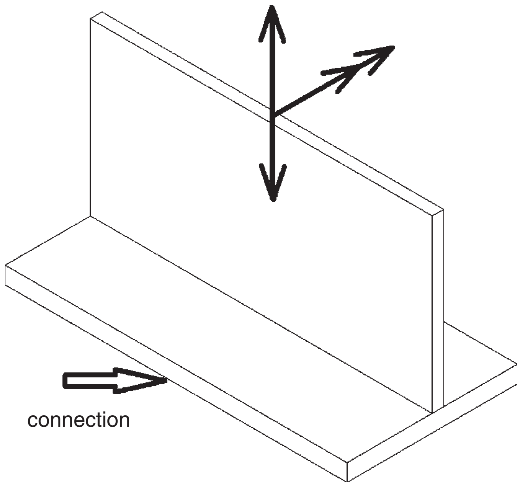

Figure 1.13 A T-stub.

The problem has no easy solution, in general. Some specific problems have been studied adding strong limitative hypotheses to geometry, and one of the most frequently used of these is a *T-stub* (Figure 1.13), which is a tee of short length connected, usually by a flange, to some other part, and loaded by a tensile or compressive axial force (force vector in the plane of the T cross-section, parallel to the web direction), or a bending moment (usually with the moment vector normal to the T-stub web). Under these simplified loads, and considering symmetrical and regular bolt "rows" connecting the flange, it is possible to assess the value of the limit plastic load, within the frame of yield line theory, also considering prying forces (see also Section 1.2.7).

In AISC 360-10, the T-stub is almost never explicitly mentioned, nor are given rules referring to its use in this specific context (but the T-stub model is explicitly used in prying force evaluation). Generally, the standard is less prescriptive and more flexible and open than the Eurocode.

In Eurocode 3, Part 1.8 the T-stub paradigm is introduced in Section 6, entitled "Structural joints connecting H or I sections", and specifically at subsections 6.2.4 "Equivalent T-stub in tension", and 6.2.5 "Equivalent T-stub in compression".

Eurocode 3, Part 1.8, Section 6.2 "Design resistance" uses the T-stub model in many of its subsections, and referring to problems that are not specifically those of a T-stub, for instance:

- §6.2.6.3(3) – "column web in transverse tension"
- §6.2.6.4(1) – "column flange in transverse bending"
- §6.2.6.5 (1), – "end-plate in bending"
- §6.2.6.6(1) – "flange cleat in bending"
- §6.2.6.8.(2) – "beam web in tension"
- §6.2.6.9(2) – "concrete in compression including grout"
- §6.2.6.10(1) – "base plate in bending under compression"
- §6.2.7.1.(8) – "extended end plate joint"
- §6.2.8.2(1) – "column bases only subjected to axial forces" (where three "non-overlapping" T-stubs are used – see Figure 1.14).

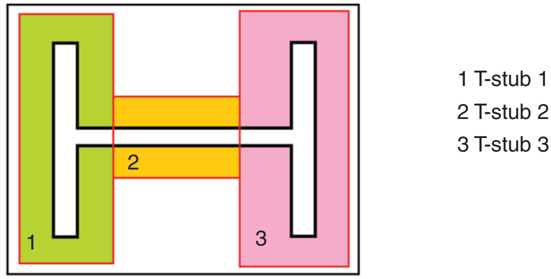

Figure 1.14 Eurocode 3, Part 1.8: column bases only subjected to axial forces.

Usually the typical sentence is that something like "…should be taken as equal to the … of equivalent T-stub representing…".

Thus, in Eurocode 3, Part 1.8, the T-stub model is used as an all-rounder to cover almost all the real-world problems of "joints connecting H or I sections". Often, practitioners have extended the scope of Eurocode 3, Section 6, to joints using L, C, T, composite, CHS, RHS, and thin-walled cross-sections, and spreadsheets using the "T-stub" paradigm can be found almost everywhere.

The author is not convinced that this approach is necessarily always the best available in 2017. This is a paradoxical result: every sub-part of every joint appears to be necessarily modeled by a T-stub, and the use of by far more general and flexible approaches such as finite elements are not even explicitly mentioned by the standards (the words "finite elements" *do not appear* in Eurocode 3, Part 1.8: is that not so?!). This is probably due to the fact that the finite element technique is still considered somehow exotic, and that it is assumed that it would take too much time to prepare and run a model. In the author's view, this is technically just not true anymore because, from experience, for many problems detailed finite element models can be prepared automatically and run in a few tens of seconds.

1.3.3 Eurocode 3 Component Model

In order to compute the stiffness and the resistance of standard connections, Eurocode 3 uses a method called the "component method" (Jaspart 1991) which decomposes connections into a number of standard components, whose uniaxial elastic stiffnesses are then evaluated by means of simplified formulae (not necessarily simple). By composing the elementary stiffnesses it is ideally possible to model the behavior of sets of these elementary components. Much excellent work has been done by following this research path, and in-depth results have been obtained for specific typologies of connections, including in the non-linear range.

The following elementary components are listed in Table 6.1 of Eurocode 3, Part 1.8 (basic joint components):

- column web panel in shear (k_1)
- column web in transverse compression (k_2)
- column web in transverse tension (k_3)
- column flange in bending (k_4)

- end plate in bending (k_5)
- flange cleat in bending (an angle cleat is assumed as example) (k_6)
- beam or column flange and web in compression (k_7)
- beam web in tension (k_8)
- plate in tension or compression (k_9)
- bolts in tension (k_{10})
- bolts in shear (k_{11})
- bolts in bearing (k_{12})
- concrete in compression including grout (k_{13})
- base plate in bending under compression (k_{14})
- base plate in bending under tension (k_{15})
- anchor bolts in tension (k_{16})
- anchor bolts in shear (k_{17})
- anchor bolts in bearing (k_{18})
- welds (k_{19})
- haunched beam (k_{20}).

Note that the same physical part may appear more than once.

A uniaxial stiffness is then related to each of these components, evaluated as a function of simplified geometry, possibly of a T-stub model, and of the loads applied. Summing up the elementary uniaxial flexibilities related to each component would then allow us to model complex connections, which can thus be seen as an assembly of these simple "bricks" in series or parallel.

Some of the stiffness related to the elementary components is considered infinite, whatever the exact geometry. Specifically the following stiffnesses are considered infinite:

- column web panel in shear, if stiffened (k_1)
- column web in compression, if stiffened (k_2)
- column web in tension, if stiffened welded connection (k_3)
- beam flange and web in compression (k_7)
- beam web in tension (k_8)
- plate in tension or compression (k_9)
- bolts in shear, if preloaded (provided that design forces do not induce slip) (k_{11})
- bolts in bearing, if preloaded (provided that design forces do not induce slip) (k_{12})
- plate in bending under compression (k_{14})
- welds (k_{19})
- haunched beams (k_{20}).

The infinity of these stiffnesses must be understood in a relative sense: these are much higher than those of the other components, and so their reciprocal (flexibility) can be assumed null.

The working mode of beam elements (or columns) is apparently reduced to axial force plus strong axis bending. This bending plus axial force causes elementary forces with a lever arm z, also suggested by the code. A 2D model is implicitly assumed and these forces will find their path to the connected part (a column in beam-to-column connections, or a concrete slab in base joints, for instance).

The rotational stiffness S_j of the joint can be obtained by the following general formula (which is 6.27 of Eurocode 3, Part 1.8, E is Young's modulus):

$$S_j = \frac{Ez^2}{\sum_i \frac{1}{k_i}}$$

where the summation is extended to all applicable components having elementary stiffness k_i.

This model *seems* simple, and the idea to decompose complex systems down to elementary ones is indeed brilliant, but how general is it, really? Neglecting part of the member forces is a strong limitation. The lever arm is problem dependent and also load dependent. On the other hand, if the geometrical simplifications that led to the k_i stiffnesses evaluations for components using T-stubs are not fully applicable, the method cannot be used.

The method is difficult to apply even for simple systems, but it is simply not applicable for complex systems. However, it is applicable to a limited set of connections. The member forces and moments should be decomposed into simple forces, acting at well defined points, where some sort of load path must be drawn. In considering this load path, the previously enumerated standard components must necessarily be found. Every possible cause of non-compliance must be neglected.

The standard states at clause 6.1.1.(1) (emphasis added):

> This section contains design methods to determine the structural properties of joints in frames of *any* type. To apply these methods, a joint should be modeled as an assembly of basic components.
>
> ©CEN, reproduced with permission

And in a "Note" at clause 6.1.1(2) (emphasis added):

> The design methods for basic joint components given in this Standard are of general application and can also be applied to similar components in other joint configurations. *However the specific design methods given for determining the design moment resistance, rotational stiffness and rotation capacity of a joint are based on an assumed distribution of internal forces for joint configurations indicated in Figure 1.2.* For other joint configurations, design methods for determining the design moment resistance, rotational stiffness and rotation capacity *should be based on appropriate assumptions for the distribution of internal forces.*
>
> ©CEN, reproduced with permission

Figure 1.2 of Eurocode 3, Part 1.8 lists the following typical joints:

- major axis: single-sided beam-to-column joint configuration
- major axis: double-sided beam-to-column joint configuration
- major axis: beam splice
- major axis: column splice

- major axis: column base
- minor axis: double-sided beam-to-column joint configuration
- minor axis: double-sided beam-to-beam joint configuration.

So it is very clear that the rules coded by Eurocode 3, Part 1.8, Section 6, refer to, and are proposed for, a limited number of joint configurations, considering I or H cross-sections, that is, they are not a general tool to deal with the problem of connection checks.

In the present context, the member stress state is general and there is no practical distinction between "major axis" bending and "minor axis" bending as it is assumed that they do act at the same time, and the same applies to torque, shears and axial force.

In this book a different path will be tried, albeit with some broad concepts in common with that of "component model".

1.3.4 Distribution of Internal Forces

As we have seen in the previous section, the detailed formulations provided by Eurocode 3, Part 1.8 in order to check connections falling in the categories delimited by the code, hold true because "appropriate assumptions for the distribution of the forces" have been used. These *appropriate distributions* are behind all methods.

In turn, these appropriate distributions are tightly related to specific configurations for which it has been found that they are useful and "appropriate". Different node layouts will surely lead to different force distributions, and the lack of a general tool to compute such distributions is one important reason why general methods to check connections were not available.

Looking at the problem in a general way, we can say that, as will be shown in Chapter 5, finding an appropriate distribution of internal forces in a general context is the main problem of connection design.

Traditional approaches use ad hoc internal force distributions that are the result not of a calculation but of a free choice which implements some interpretation of connection experimental behavior. This can be done with the hope of being right if and only if the lower bound theorem of limit analysis is applicable. The use of somewhat arbitrary but balanced force configurations is behind all the traditional approaches as well as some of the new ones, and is also behind the success of many historic buildings. As Prof. Jacques Heyman has clearly shown in his marvelous book *The Stone Skeleton* (Heyman 1995) without the lower bound theorem no design could be carried out, as the "true" distribution of internal forces is basically impossible to determine. And this is also true for steel connections.

This is a key concept, and one of the main pillars to be considered in connection checks.

1.3.5 Prying Forces

Prying forces are the forces that arise at the contact between plates due to the flexure of the plates themselves induced by bolt tensile forces. They are statically undetermined, and strongly depend on the loads applied, on the geometrical configuration of the plates in contact, on their thicknesses, and on the position and diameter of the bolts.

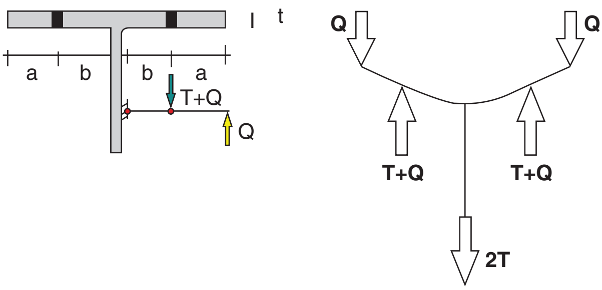

Figure 1.15 Prying forces *Q* increase tensile forces in bolts *T*. Simple T-stub model.

Prying forces imply an increase in the bolt tensile forces, and so must be taken into account in order to avoid the bolt tensile forces being underestimated. On the other hand, prying forces unload those plates that find, in the contact to another plate, a useful bearing of a free tip.

The widely used method of evaluating prying forces is due to Thornton (1985) and refers to a T-stub model (Figure 1.15). If the geometry is different, the method cannot be applied. Other simple models can be used instead (e.g. cantilevers or simply supported "beams"), but with questionable reliability. At the moment, the evaluation of prying forces is one of the most complex problems of generic steel connection analysis and can only be tackled for generic geometries (i.e. not T-stubs), by means of contact non-linearity and finite elements; the only possible way to better compute prying forces in a general context is to use plate–shell finite element models and consider contact non-linearity. This is a quite complex issue but, as we will see, it can be tackled by proper finite element analyses run automatically.

1.3.6 Block Tearing

Block tearing is the fracture and subsequent separation of a part of a steel plate, usually in a bolted connection, under the effect of the shear forces transferred to the plate by the bolt shafts.

In the available coded standards, block tearing is dealt with by considering shear stress paths and normal stress paths, mixed together so as to define cut lines for the plate at hand. Usually, also in the textbooks, it is apparently assumed that the rupture lines are aligned with plate sides (Figure 1.16), so that the geometry of the fracture lines is somehow forced to respect the external geometry of the constituent. However, this is not true in general. The forces transferred from the bolt shafts to the plate at hand are not only generally different from each other, but also have different inclinations. So, once again, it appears that the methods usually adopted to consider this failure mode are by far too simplified, and that a more general model should be used to properly evaluate this dangerous condition.

1.4 Scope of This Book

The aim of this book is to discuss the problems to be faced when trying to tackle steel connections analysis in a general way.

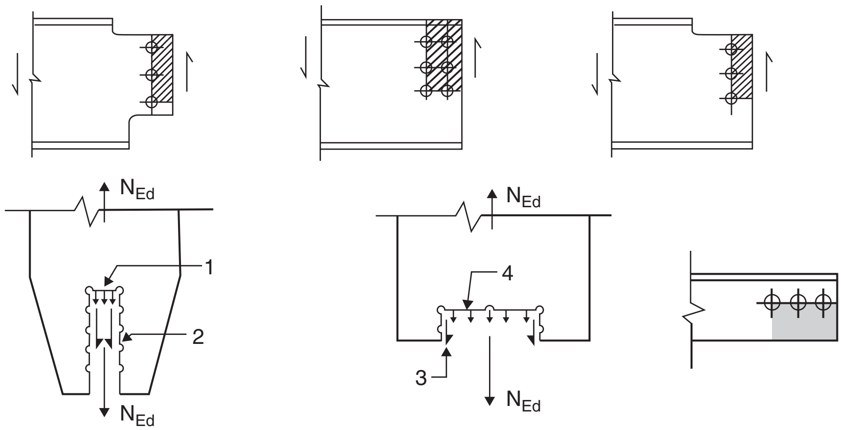

Figure 1.16 Block tearing (from Eurocode 3, Part 1.8): forces are aligned with plate sides, and so with the break lines. ©CEN, reproduced with permission.

The chapters will broadly follow the path of the author's research, starting from a generic 3D finite element model of a steel structure, like the thousands that are created every day by structural engineers all around the world. This is a model using may be hundreds of beam and truss elements, possibly inclined, and using generic cross-sections, tens of load cases, and many load combinations, and has probably been run in linear or non-linear range to check members according to some available standards (i.e. neglecting the problem of connections, and considering the members as wireframe sticks to be checked for member failure modes: resistance, stability, and deformability).

Now, the problem is to check connections according to the computed member forces, or according to the capacity design rules. The structure is 3D, members are inclined, so a first question is: how many different connections are there, and what stress levels are expected?

Then, once these different connections have been detected, it is necessary to describe how they are physically realized, one by one. So, it is necessary to set up what in this book is called the *scene* for each node. Finally, all the constituents, members, cleats, and connectors must be checked against all possible failure modes, and taking into account all load combinations applicable.

This is the task to be accomplished. The chapters of this book deal with this task.

Chapter 2 deals with *jnodes*, that is the connections as they can be described in a wireframe context. This chapter will explain how to detect equal jnodes automatically, and how to classify them (the so-called *jnode analytics*).

Chapter 3 is short and proposes a general model for connections. This is needed for the next steps. This chapter also introduces the concept of the connection graph.

Chapter 4 is related to *renodes*, the 3D counterpart of jnodes. Here every constituent is a 3D object in space, with a specific position, orientation, and shape. This chapter also deals with the problem of automatic detection of "connections", and automatically analyzing the coherence of the node. The *chain* concept will also be introduced here.

Chapter 5 is a review of the general principles that are always applicable and that must be complied with for a sound analysis of connections. They will be extensively used in the remaining chapters of this book. Also outlined here is the statics of connections, introducing iso-, hyper-, and hypo- connected nodes.

Chapters 6 and 7 are related to *connectors*, which are the devices used to physically implement a connection. Chapter 6 deals with welds, and Chapter 7 with bolts and contact. The aim is to review their mechanical behavior, the computational methods used to model them, and the hypotheses currently adopted by the methods in use.

Chapter 8 is a discussion of the most important failure modes that need to be checked in connection design, and of some of the models that can be used to check them, with particular reference to AISC 360-10 and Eurocode 3, Part 1.8. A more general method for block-shear is presented.

Chapter 9 deals with a first general method proposed to analyze connections: the *hybrid approach*, using several techniques to get quick results without merely relying on pure computational force. This basically explains how to check connections using a general method.

Chapter 10 refers to a pure fem approach, that is, the ideally unified, general, and single method that could be used to check all connections: it is the future for this subject, and is still being researched.

Chapter 11 draws some conclusions and discusses the open problems.

Finally, the appendices deal with specific problems such as tangent stiffness matrices, and with notation and symbols.

For the most part the proposed and discussed method has already been implemented, tested, and used by engineers, by running the software developed by the author and called CSE (Connection Study Environment).

1.5 Automatic Modeling and Analysis of 3D Connections

Times are changing: traditional approaches to steel connection design and checking will assume a secondary role, while the use of general methods, equivalent to those already used dealing with *structures*, will increase in importance and, gradually, will become the normal way to check connections. As has already happened with 3D framed structures, traditional approaches will still retain their value as cross-checking or initial sizing tools.

The computational power available nowadays makes it possible to create and run a finite element model in a matter of a few tens of seconds, and so, at least for some of the failure modes to be investigated, this way of checking is today the most efficient and promising. The limitations of traditional approaches to steel connection checks seem to be particularly evident when they are used together with modern computational tools, that is, the finite element software broadly used to model frames and structures by means of beam and truss elements. It is evident, then, that there is a clash between the need for using hundreds elements in a 3D context, loaded by tens or even hundreds combinations, and the oversimplified, much-throwing-apart, simple methods that forget to see the connections for what they are: complex structural subsystems that need general methods to be analyzed.

1.6 Acknowledgments

This book is the result of an individual research program carried on since 1999, with the aim of providing a general software tool able to check *generic* steel connections. This software tool has been written step by step, and has moved forward together with the results obtained by research. It is referred to as the *Connection Study Environment*, and has been on the market since 2008 (www.steelchecks.com/CSE). All the computational results presented in this book come from using CSE, and all the connection models in the figures have been modeled using CSE. The reader interested in a brief history of the project and the reasons why it was carried on, will find this information in the final Chapter 11.

The reason why this work was carried on by a single researcher is readily explained: none of the possible partners in Italy seemed interested, 10–15 years ago, in spending years of effort with the aim of developing a general tool dealing with steel connections. Some very important early results were presented at a National Conference on Steel Structures in Padova (Rugarli 2009), but at the time, nobody was interested – the project seemed too exotic.

The author wishes to thank, with deep feelings of gratitude, his unforgotten Professors at the Politecnico di Milano, and particularly Francesca Rolandi, Adelina Tarsi, and Laura Gotusso, who taught mathematical analysis and numerical analysis, Carlo Cercignani, who taught rational mechanics, Leo Finzi who taught elasticity, Giulio Ballio who taught steel structures, Leone Corradi dell'Acqua who taught instability, and Giulio Maier, who taught elasto-plasticity and finite elements during an unforgettable and life-changing year, some 33 years ago.

The author also wishes to thank all the colleagues around the world, of different cultures, languages, and origins, something marvelous in itself, who have trusted the work done during those years, and allowed it to be carried on, using the software procedures that, from year to year, have been developed and improved. This is what sometimes is reductively called *the market*. With some of them, the discussions were frequent and/or very useful, so thanks in particular to Emanuele Alborghetti, Giovanni Cannavò, Michele Capè, Marco Croci (who also helped greatly in developing tests and parameterizations and spent several years working on this project), Jason McCool, Andy Gleaves, Meda Raveendra Reddy, Sergio Saiz, Christos Saouridis and the colleagues at CCS, Harri Siebert, Edoardo Soncini.

While every effort has been made to deliver a text with no misprints or errors, it would be wise for the reader to check at: http://steelchecks.com/connections/SCA.asp to see whether any *errata corrige* is available, and also to be informed about the latest results of the ongoing research.

Finally, the author wishes to deeply thank Paul Beverley for his precious, careful, and skilled work in editing this book.

References

Booth, E. 2014, *Earthquake Design Practice for Buildings*, 3rd ed. ICE Publishing, London.

Bruneau, M., Uang, C.M., and Sabelli, R. 2011, *Ductile Design of Steel Structures*, 2nd ed., McGraw Hill.

European Spreadsheet Risk Interest Group, www.eusprig.org.

Fraldi, M., Nunziante, L., Gesualdo, A., and Guarracino, F. 2010, *On the Bounding of Limit Multipliers for Combined Loading*, *Proc. R. Soc., A* 2010, **466**, 493–514.

Heyman, J. 1995, *The Stone Skeleton*, Cambridge University Press, Cambridge.

Huckelbridge, A.A., Palmer, D.A., and Snyder, R.E. 1997, *Grand Gusset Failure*, Civil Engineering, Vol. 67, September, pp. 50–52.

Jaspart, J.P. 1991, *Etude de la semi rigidité des nœuds poutre-colonne et son influence sur la résistance et la stabilité des ossatures en acier*, PhD Thesis, Department MSM, University of Liège.

Kemeny, J. 1979, *The Need for Change: The Legacy of TMI*. Report of the President's Commission on the Accident at Three Mile Island, Government Printing Office, Washington DC.

Joints in Steel Construction – Moment Connections, P207/95, 1995, The Steel Construction Institute, ("Green Book").

Reason, J. 1990, *Human Error*, Cambridge University Press, 1990.

Rugarli, P. 2004, *Combinazioni di Verifica allo Stato Limite: il Non Detto delle Normative, Ingegneria Sismica*, **2**, 2004.

Rugarli, P. 2009, *Software Computation of Steel Joints Built Up Freely Placing Components* (Italian), Acta XXII CTA Congress, L'Acciaio per un Futuro Sostenibile, Padova September 28–30, 2009.

Rugarli, P. 2014, *Validazione Strutturale*, EPC Libri, Rome.

Thornton, W.A. 1985, *Prying Action – a General Treatment*, Engineering Journal, American Institute of Steel Construction, Second Quarter 1985.

White, D.W., Leon, R.T., Kim, Y.D., Mentes Y., and Bhuyan, M. 2013, *Finite Element Simulation and Assessment of the Strength Behavior of Riveted and Bolted Gusset-Plate Connections in Steel Truss Bridges – Final Report*, Georgia Tech, 2013.

2

Jnodes

2.1 BFEM

The most commonly used finite element models are those made up by beam and truss elements. These models are used to design, analyze, and check a wide range of structures, such as buildings, towers, pipe racks, industrial plants, roofs, and bridges.

Although not all designers seem aware of the fact, the wireframe models made up using beams and trusses are finite element models like those made up by more complex elements, like plate–shell elements, membrane elements, and solid elements.

Starting from the mid 1980s – not long ago from a historic perspective – engineers and architects modeled structures using a computer, and prepared 2D models and then, a few years later, 3D models. By the end of the 1980s and into the 1990s it was common to see 3D wireframe models of structures, having thousands degrees of freedom, solved in the elastic range for perhaps tens of load cases.

This was brand new: never before had so many engineers been able to create such complex models. Also, the requirements of the standards grew quickly. If at the end of the 1970s it was common to check structures for a few, unrealistic envelope combinations, by the mid 1990s it was common to study finite element models of 3D structures using tens of load cases and combinations. This opened a quite new scenario, which is still under research: how to use the computing power of personal computers to deal with complex analytical problems, with no loss of safety or comprehension by the analyst. Elsewhere (e.g. Rugarli 2014) it has been pointed out and discussed in depth that the validation of finite element models is a complex problem, involving several disciplines, and it is also an urgent problem as too often these models are unchecked by their creators and lead to completely wrong analyses.

These finite element models were usually based on the two most simple finite elements available: the *truss element*, modeling members carrying only an axial force, and the *beam element*, an element capable of absorbing one axial force, two shears, one torque and two bending moments, all variable along its axis. There are also curved elements, but by far the most frequently used elements were and are straight, using two *nodes*.

As I have tried to explain elsewhere in more detail (Rugarli 2010), a *finite element* is a portion of a continuum, governed by a set of usually complex differential equations, having a very simple shape. The shape of the finite element is fully described by a limited number of points that in the finite element method (fem) jargon are called *nodes*. The basic idea of fem is that what happens inside the finite element is entirely dependent

Steel Connection Analysis, First Edition. Paolo Rugarli.

on the displacements of the nodes, which are the main problem unknowns (there are also fem approaches based on forces, but these are less common).

The structure as a whole is divided into finite elements, and the displacements of all the *nodes* form the unknown vector of the problem to be solved. From the nodal displacements, once known, we can derive the strains internal to the elements, and from the strains the stresses. This is done by means of simple interpolation formulae, usually polynomial: the polynomial functions used to interpolate are fixed a priori, and the displacements of the nodes are their weights. If the finite elements are small, when compared to stress gradient, then results are near the ones which can be obtained by solving the partial differential equations by other methods, such as finite differences, closed formulae, or series.

One important feature of beam and truss elements is that in elasticity problems their polynomial interpolation is exact if no internal loads are applied to the elements themselves: a linear variation of displacement is exact for a pulled bar, and a cubic displacement variation is exact for a Bernoulli beam in flexure without intermediate loads. This last can be seen by remembering the governing differential equation $EJw^{IV} = 0$, with E Young's modulus, J second area moment, and w^{IV} the fourth derivative of $w(x)$ to x. This means that beam and truss elements do not need to be very small, because their polynomial interpolation is exact, or very nearly exact (the additional effect of the internal loads can be re-added by a simple sum to the results acquired by interpolation).

Therefore, the subdivision of the structure in finite elements is easy: very often, a member is modeled with just one finite element. This explains why so many practitioners are not aware that they are indeed using finite elements, because they think they are using "bar" or "member" elements.

In non-linear analysis, each increment of displacement is related to an increment of strains, and that to an increment of stresses. Final stresses are obtained by summing up increments. However, in non-linear analysis, more refined meshes are needed for beam elements, and so in that context the need for small finite elements, typical of plate and membrane structural elements, or of solid elements, is also applicable to beam elements.

The starting model for designing a steel structure made by 1D members is thus a finite element model mainly made by truss and beam elements. In this book there will be several different types of finite element models, so it is necessary to distinguish them by a proper nomenclature. For this reason, this starting finite element model is named here the *BFEM*, where the "B" stands for beam or bar – or also for Bernoulli.

BFEMs are used very frequently in today's engineering practice. Sometimes they are not made *only* by beam and truss elements. For instance, in modeling a building, the bracing concrete core or shear walls may well be modeled by membrane or plate–shell elements, and also the reinforced concrete slabs. However, this model will always embed many beam and truss elements, which are currently used to model structural elements such as columns, beams, and bracings.

The existence of beam and truss elements is needed to properly model the skeleton of the structure, and after solving these elements are usually checked against pertinent standards, such as Eurocode 3 (EN 1993), AISC (AISC 360-10), or British Standard 5950, in order to test their proper design. This is the starting point for the most part of the steel structures, which are the object of this book.

Note that, when considering the problem of steel connection analysis, dealing with the familiar internal forces called “axial force”, “bending moment”, “shear”, or “torque”, the existence of an underlying model consisting of 1D structural elements is implicitly assumed. These follow the well known basic structural theories related to beam and truss elements.

Although it is not necessarily true that the original computational model is a finite element model, i.e. a BFEM, this is the most likely situation nowadays, and it is always possible to describe a structural layout, also perhaps computed by hand using virtual work principle, by means of a BFEM.

2.2 From the BFEM to the Member Model

2.2.1 Physical Model and the Analytical Model

In this book the term *member* refers to a component of a steel structure, usually straight and prismatic, but also possibly tapered or curved, which can be modeled by means of 1D structural theories (Euler–Bernoulli’s beam theory, Timoshenko’s beam theory, etc.). A member is fabricated as a unique piece and may possibly be connected to other members or to other structural elements such as plinths, concrete slabs, and walls, by means of fittings and connectors such as bolts and welds.

So, connection analysis is the analysis of the connection between members, and between members and other structural elements. It is easy to also conceive connections between structural elements which cannot be modeled by means of 1D structural theory, but these are not covered in this book: e.g. the connection between a plate and a wall or the connection of a tensile membrane to a ring.

The connections that are going to be considered here always involve at least one member, and refer to a limited part of its axis: usually the extremity, but also possibly, as will be seen, a limited internal part of its internal length (these will be named *passing members*).

When a steel structure is analyzed in order to compute internal forces and displacements under the applied loads, the analytical model is usually a finite element model. However, the physical model, which takes into account the members, is a different one, purely geometric.

Some software programs ask the users to prepare a physical model, which is later converted into an analytical model by adding the necessary finite elements. Some other programs do not use physical models, but only analytical models. If the need is only to analyze a structure in order to compute internal stresses, the physical model is not needed. Therefore, there are programs that deal with finite elements and do not have the *member* concept.

In order to hide the finite element model details, some software is designed so that the finite elements are kept in the background, possibly generated automatically according to internal rules. If this is the case, the user of the software has direct input *members*, and the finite element model has been generated by specific subsequent commands.

The automatic creation of a finite element model from a member model is not an easy matter and there are specific problems related to rigid offsets, releases, connectivity and so on. The danger of such automatic generation is that it can create finite element models that are hard to check and that the users have to trust with no direct control over them.

Sometimes it is then far better to generate the analytical model directly or, if a physical model is the starting point, to generate the analytical model with a step-by-step controlled procedure. For this reason, many design software programs are based on the analytical model, that is, they manage finite elements.

The advantage of this approach is that the expert analyst directly controls the analytical model, and is responsible for its meaningfulness. Usually this approach is preferred by structural engineers, expert in analysis, especially when the structures are complex; checking an analytical model created automatically from a drawing or from a physical model can be a nightmare, and the time necessary to clean up the fem may be much longer than the time needed to rebuild the analytical model from scratch.

Moreover, sometimes the rules used to generate the finite element model from the physical model or from the drawing cannot be shared by the expert analyst, as they introduce violations of basic concepts, such as keeping the axes line in the centers of the cross-sections, modeling properly the eccentricities, neglecting those unneeded, and so on. In many engineers' experience, it is rare that such automatic transforming of a physical model or of a drawing into a finite element model leads to sound and checked models. Very often, the models are trusted as such, and no real control is exerted, as it would be too expensive.

If the member model is already available, and a BFEM is not available, in order to properly consider the internal forces which try to detach a member from its neighbors, an underlying analytical model must be set up. Otherwise it is just not possible to properly assign forces to members, and to properly take into account the points where these forces are exchanged. This is not always well understood when a simple drawing of physical members is used to compute connections. To analyze connections an underlying analytical model is needed.

If a simple beam to column connection is considered (e.g. Figure 2.1), in order to check the connection several questions must be answered, and the pure geometric position of the parts (the plate, its thickness, the welds, and the bolts) is not enough to answer to such questions:

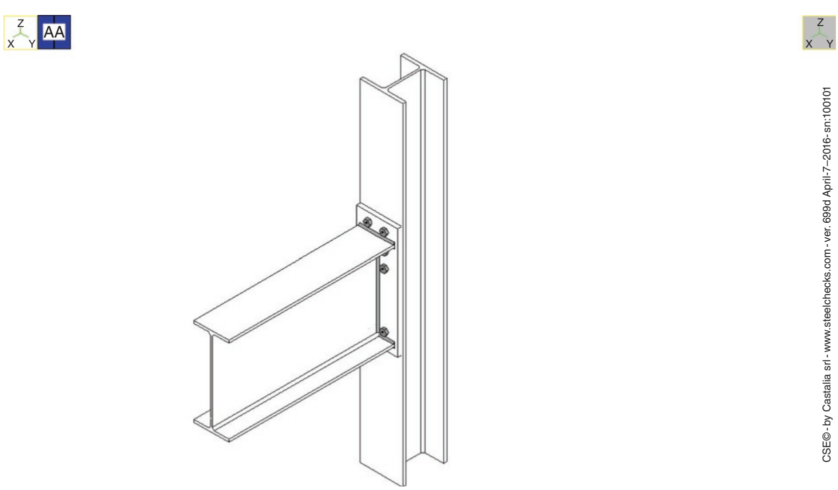

Figure 2.1 A beam to column connection.

1) What are the internal forces being exchanged? Is it just shear or also bending moment(s) and axial force and torque?
2) Where are these forces exactly exchanged, i.e. at which exact points? This is important because forces trigger moments, far from their line of action.
3) Have *the members* been checked for resistance and stability *coherently with these points of forces exchanged*?

So, willingly or not, a physical or geometrical model used to check connections is always supported by an underlying analytical model, which can be coherent or not with the analytical model used to check members. Ignoring the need for such coherence is a possible problem, as some basic principles are violated: the force distribution used to check connections is not the same force distribution used to check members, which violates the static theorem of limit analysis, as will be seen in Chapter 5. Sometimes this violation is done consciously, but other times not. Some people, starting from a drawing and being willing to check connections, are apparently unaware of the problem.

If, on the other hand, the analytical model is the only one available, a first necessary step in order to analyze connections between members is to convert it into a physical model by finding the members. This task could ideally be accomplished with direct selection of elements and then assigning them to members, but when dealing with a complex 3D structure this process may be long and cumbersome.

Therefore, if a BFEM has been prepared, the analyst needs to automatically convert a finite element model made by beam and truss elements into a member model.

2.2.2 Member Detection: Connection Codes

If the members are straight and prismatic, as usually happens, they can be detected by an automatic search.

The following will be assumed:

- A single truss element is always a member.

These elements are typically used for bracings.

If a set of n beam elements is such that:

- all the finite elements are connected one to one so that they can be ordered in such a way that element i is connected to element $(i + 1)$ in a node N_{i+1}
- all the finite elements are aligned between themselves (each element i is aligned to every element $j \neq i$) within a given tolerance
- all the finite elements have the same cross-section
- all the finite elements have the same material
- all the finite elements have the same orientation in space
- there is no end release applied to any element extremity connected to an internal node (that is a node N_k with $2 \leq k \leq n$)

then, it is assumed that this set of aligned beam elements is modeling a straight prismatic member, which is by far the most frequent type of member in steel structures.

Both the internal and the external nodes can be connected to other elements, which are part of other members; the nodes where this condition is met will be the place of connections between members.

When considering the alignment of two elements, say *Element i*, and *Element j*, the following considerations apply.

Let N_1 and N_2 be the nodes of *Element i*, and N_3 and N_4 be the nodes of *Element j*. If *Element j* is adjacent to *Element i*, then $N_2 = N_3$.

If no rigid offsets have been used to define beam elements, then the alignment condition will refer to the set of three 3D points (N_1, $N_2 = N_3$, N_4), in space.

On the other hand, if rigid offsets have been defined, we must distinguish between *element node* and *element extremity*.

The *element node* is the 3D point whose displacements are the primary unknowns in the displacement-based finite element method.

Element extremity is a 3D point obtained by adding to the node position the rigid offset. This point marks the end of the deformable part of the element axis. Extremity points can be named "true end points", as they are the true deformable element extremity points. If no rigid offset is applied, they collapse to nodes. Generally speaking *a node* of a beam element is not coincident with its *extremity*.

Let N_{T1}, N_{T2}, N_{T3}, and N_{T4} be the *true end points* of *Element i* and *Element j*. To consider aligned adjacent elements *Element i* and *Element j* the following conditions must be met (see Figure 2.2):

- N_{T2} must be coincident with N_{T3}.
- (N_{T1}, $N_{T2} = N_{T3}$, N_{T4}) must be three 3D points aligned in space.

Considering the three sets of elements in Figure 2.3, and assuming that all the elements have the same material and cross-section, the following considerations can be used (ordering of elements is from left to right):

- Elements of set A cannot be a member, as there is one end release applied to the second element from the left.
- Elements of set B cannot be a (straight) member as they are not aligned.

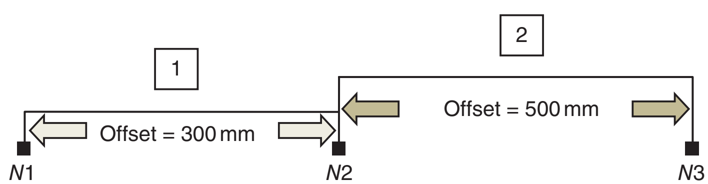

Figure 2.2 Example of two adjacent elements whose nodes are aligned but that, due to the existence of rigid offsets, cannot be considered aligned.

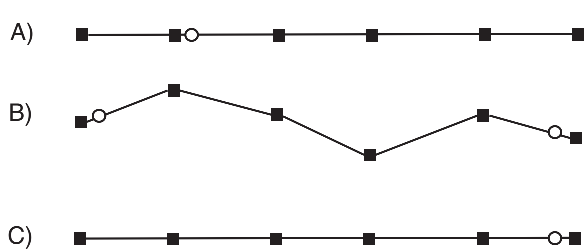

Figure 2.3 Sets of finite elements.

- Elements of set C can be a member, as the end release is applied to the end element second extremity. However, all the elements will have to be equally oriented.

By definition, *a member has no internal release.*

Given a set of aligned beam elements, possibly mapping a member, it is assumed that if an internal end release is found, no matter which component(s) are released, then this set of *elements* is divided into two straight *members* by the end release found. If the number of *internal* extremities with release applied is n, then there will be at least $(n + 1)$ members.

If a discontinuity of material, cross-section, or orientation is detected then the set is split, and it will give rise to more members.

This simple set of rules is able to automatically detect most of the members in an analytical model, but there are some specific cases, which must be dealt with.

1) Two aligned equal members may be connected in such a way that no release is to be applied to the underlying beam elements: a splice joint between two identical members is the typical example (Figure 2.4). Therefore we need the ability to split it into two members, equal beam elements aligned, with no end release inside and equally oriented, if needed.
2) A member may be tapered.
3) A member may be curved.

The first issue is solved considering *connection codes*. It will be assumed that a connection code can be optionally assigned to the extremities of beam elements.

If a connection code is assigned to a beam element extremity, then that extremity is also the end of the member to which the beam element belongs.

If the connection code is not assigned to the beam extremity, then the member to which the beam element belongs can be extended to more (aligned and member-compliant) beam elements, if any.

Besides, if an end release of any kind is applied to a beam element extremity this automatically implies a connection code. However, a connection code may be applied with no end release.

Connection codes are not usually dealt with in standard finite element software. However, they can be easily added or mapped to groups, as follows. If a software program deals with the concept of group or layer, three special groups or layers can be set up:

- group/layer “CCI”, whose elements are beam elements having a connection code at I node (first node)

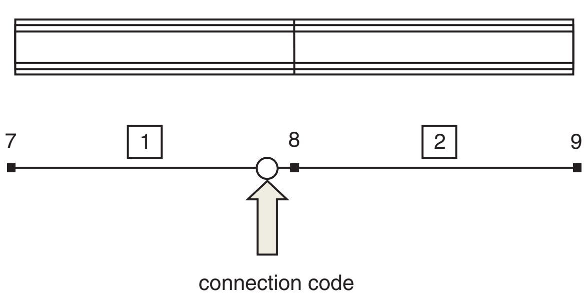

Figure 2.4 Aligned elements belonging to different members: no end release.

- group/layer "CCJ", whose elements are beam elements having a connection code at J node (second node)
- group/layer "CCICCJ", whose elements are beam elements having a connection code both at I and J extremities

Elements having end releases, by definition, have a connection code at the end released, so there is no need to add them to the groups/layers mentioned.

In standard finite element software, it is easy to add connection code management. The mask used to choose the end releases, may be of seven Boolean flags instead of six. The seventh code would be the connection code.

- If one end release is applied, then automatically the seventh flag (connection code) is switched on.
- However, if no end release is applied, the user might set the seventh flag on, without affecting the releases of the beam element.

This procedure has been applied in the code developed by the author (Sargon©), with excellent results. In view of special problems (Section 2.7.2) it is also possible to assign a color to a connection code.

Getting back to the specific cases listed, the second issue (tapered members) can be dealt with by appropriate checks on the element cross-sections, provided that single beam elements can be tapered. If this holds true, then the beam element will have two different cross-sections at the two extremities, and the conditions to check will also have to include that no abrupt change of cross-section may happen at member–element interfaces. For linearly tapered members, a more strict condition can be that the slope from one finite element to the adjacent one does not change.

The third issue (curved members) can be managed by releasing the alignment tolerance, that is, assuming that two straight adjacent elements may have a small difference in alignment, and still be considered part of a single member.

2.2.3 An Automatic Algorithm for Straight Prismatic Member Detection

A brief pseudocode description of the algorithm needed to automatically detect straight prismatic members will be now given.

```
For (each Truss Element i)
{
   AddMember (TrussElement i);
}
UnselectAllBeamElements ();
For (each unselected Beam Element i)
{
   Select (Beam i);
   FindAllElementsAlignedContinuous (Set1, i, 1); // Set1 is a set
   of elements empty
   FindAllElementsAlignedContinuous (Set2, i, 2); // Set2 is a set
   of elements empty
   AddMember (Set1 + Beam i + Set2)
}
```

```
FindAllElementsAlignedContinuous(Elements, Beam n,
Extremity ext)
{
    If((Beam n).HasConnectionCode(ext) = = TRUE) return; //
    connection code reached, end

    Node = (Beam n).GetNode(ext);
    For(each unselected Beam i)
    {
        For(iext = 1, 2)
        {
            If((Beam i).GetNode(iext) = = Node) // connected I the
            same node
            {
              Point3D p1 = (Beam n).GetTrueEndPoint(ext); // beam n
              extremity
              Point3D p2 = (Beam i).GetTrueEndPoint(iext); // beam
              i extremity
              If(Distance(p1, p2) < sometolerance)  // true points
              coincide at connection
              {
                If(AreAligned(Beam n, Beam i)) // alignment
                condition
                {
                  If(GetCrossSection(Beam n) = = GetCrossSection
                  (Beam i))
                  {
                    If(GetMaterial(Beam n) = = GetMaterial
                    (Beam i)) // Equal material condition
                    {
                      If((Beam i).HasConnectionCode(iext) =
                      = FALSE)
                      {
                      Select(Beam i);
                      Elements.Add(Beam i);
                      If(iext = = 1) otherext = 2;
                      Else      otherext = 1;
                      FindAllElementsAlignedContinuous
                      (Elements, i, otherext); // recursive
                      return;
                      }
                    }
                  }
                }
              }
            }
        }
    }
}
```

This algorithm can be further improved by considering tapered members or curved members, but this is left to the reader.

At the end of the search, a set of members will be found. Each member will have:

- two end nodes, I and J (N_I and N_J)
- a single truss element, having a material, a cross-section, and an orientation. Or
- a set of underlying beam elements, ordered in such a way that the first element first node will be the first node of the member, and that the last element second node will be the end node of the member; moreover, the second node of *Element i* will be equal to first node of *Element* $(i + 1)$
- a material, equal for all the beam elements
- a cross-section, equal for all the beam elements
- an orientation, equal for all the beam elements (see Appendix 1).

2.2.4 Member Data Structure

The data structure for a member should include:

- the end-nodes identifiers
- the cross-section of the member
- the material of the member
- a flag indicating if the member is a truss or a beam
- the vector of the identifiers (numbers) of the finite elements of which it is composed
- the rigid offset relative to the nodes at the member ends
- the connection codes and the end releases of the elements at the ends
- the orientation of the member.

The orientation (Appendix 1) will be defined as the set of three unity vectors:

- The first vector, $\mathbf{v_1}$, is the member axis vector, from first to second extremity.
- The second vector, $\mathbf{v_2}$, is the strong principal axis of the cross-section.
- The third vector, $\mathbf{v_3}$, is the weak principal axis of the cross-section.

The definition of "strong" and "weak" axis is notional. For I, H, T, [,][, and similar cross-sections, it will be assumed that the "strong axis", axis 2, is normal to the web, and parallel to flanges. What is important, however, is just that a well defined rule to name principal axes is set up, for each cross-section type.

The set of the three unit (column) vectors $\mathbf{v_i}$ is able to define, by row as $\mathbf{v_i}^\mathbf{T}$, a 3×3 square matrix $\mathbf{T}$, which is orthogonal, and is such that its transpose $\mathbf{T}^\mathbf{T}$, multiplied by $\mathbf{T}$, leads to unit matrix $\mathbf{I}$: thus $\mathbf{T}^\mathbf{T}\mathbf{T} = \mathbf{I}$.

Once the members have been searched, a global vector of members will be available.

2.2.5 Member Classification at a Node

Considering a node of a BFEM, and a member, the following classification can be set up.

If the node is not connected to any elements of those defining the member, the node is *unconnected* to the member, and the member is unconnected to the node.

If the node is one of the internal nodes of the member, then the node is *connected* to the member, and the member will be classified as *passing*, at the node considered

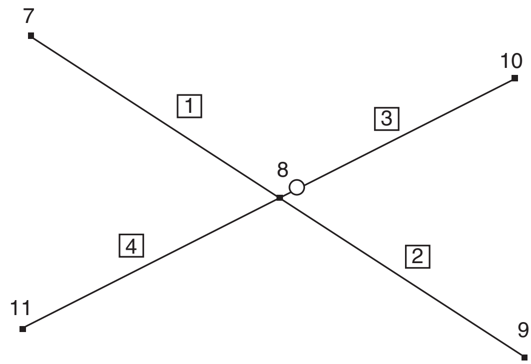

Figure 2.5 Member at a node. The member made up of beam elements 1 and 2 is passing at node 8. The member made of beam element 4 is cuspidal at node 8. The member made of element 3 is interrupted and unreleased. The full circle stands for "connection code". The small full squares mark the node positions.

(Figure 2.5). A passing member is by definition always connected, otherwise it will be considered unconnected to the node.

If a node is one of the two end nodes of the member, the member is connected to it, and connection codes must be considered. A member may or may not have, at each extremity, a connection code. The algorithm described at the previous section will end searching for more elements of a member, particularly if there is no connection code at the extremity. This will happen if:

- a discontinuity of alignment, material, cross-section or orientation is found
- no further beam element is connected to the end considered.

If the member *has a connection code* at one end, it is said to be *interrupted* at that end, or interrupted at a node (Figure 2.5). An interrupted member at a node is always connected to the node. Members interrupted at one end may or may not have there an end release. If they have one end release, they are said to be *released*, and cannot be, by definition, fully resistant members.

If a member *has no connection code* at one end, it is said to be *cuspidal* at that end, or cuspidal at a node. The word "cuspidal" is semantically related to the existence of a cusp, as the axis line comes to possibly meet other structural elements with no modification or smoothing ("connection"). As will soon be seen, such members are natural candidates for being the main member of a set of connected members.

Summing up, a member at a node can be:

- unconnected
- connected, divided into:
  - passing
  - cuspidal
  - interrupted, divided into:
    - released or
    - unreleased.

2.2.6 Member Mutual Alignment Coding

A useful classification can be set up when considering two members meeting at a node, depending on the mutual alignments of their axes (member axis and cross-section