MASTER

Automatic procedure to include imperfections in non-linearly analysed steel frames

Diallo, A.O.

Award date:
1999

Department of Structural Design

Eindhoven University of Technology

Automatic Procedure to Include Imperfections in Non-Linearly Analyzed Steel Frames

A.O. Diallo
September 1999

TUE/CCO/99.21

Supervisors:
Prof. Ir. H.H. Snijder
Dr. Ir. J.C.D. Hoenderkamp
J. Faure
Prof. J.-P. Muzeau

ACKNOWLEDGEMENT

I would like to thank everyone who helped me during my staying at the engineering company INGEROP-Clermont-Ferrand, particularly:

Mr. J.J. Yétérian, Director, for letting me doing my six month practical training within his company.

Mr. J. Faure, joint-Director, of the Metallic Constructions department, who guided me in the different phases of my project.

Prof. J.-P. Muzeau, professor at C.U.S.T., who allowed me to come and do my final graduate thesis in France.

Finally I would like to thank, Prof. Ir. H.H. Snijder, Professor at Eindhoven University of Technology, and Dr. Ir. J.C.D Hoenderkamp, Associate Professor at Eindhoven University of Technology for the realisation of this report.

ABSTRACT

Eurocode 3 allows the possibility to do a second-order analysis of steel structures. Geometrical imperfections are described in Eurocode 3 in such a way that non-linear analysis of structures can be done. Non-linear calculations necessitate the use of computer programs. Basic examples show that the application of Eurocode 3 in practical cases has some limits.

The goal of this report is to limit the problems with the "imperfections of a structure". An automatic procedure is proposed to take geometrical imperfections into account.

The automatic procedure is based on Eurocode 3. The structure is considered without imperfections but with intermediate nodes. A linear analysis of the structure allows the determination of forces and moments. A buckling analysis allows the determination of the critical buckling coefficients corresponding to several buckling modes. Based on the buckling shapes, corresponding to the buckling coefficients, it is possible to define the lowest buckling coefficient α_{CR} corresponding to the sway deformation of the structure. This α_{CR} determines whether or not it is necessary to take imperfections into account. Doing a non-linear analysis of the structure implies to take imperfections into account. To do so, frame and member imperfections need to be defined. With the help of the buckling shape, corresponding to sway deformations, these imperfections are defined. The non-linear analysis gives adjusted forces and moments; only the strength of the structure must be verified according to Eurocode 3.

This report is subdivided into two parts. First the necessary information concerning Eurocode 3 and non-linearity are described. Based on some basic examples the do's and don'ts are shown. They lead to two main problems arising in case of an automatic consideration of imperfections: the selection of the critical buckling coefficient α_{CR} and the way to apply frame imperfections. The second part suggests a way to solve those two problems and proposes an automatic procedure. The suggested idea is based on the shape of the buckling deformation resulting from an elastic buckling analysis. Finally, by two examples, an individual structural element and a single frame, the automatic procedure is verified.

The 7 – steps of the suggested automatic procedure are represented by the organisation chart of Figure 1 and go as follows:

<u>Step 1</u>: Modelling structure
Make a model of the structure with intermediate nodes.
<u>Step 2</u>: Buckling analysis
A linear analysis determines forces and moments of the structure. A buckling analysis gives the α_{CR} values corresponding to buckling modes. Select the lowest positive α_{CR} value for sway buckling.
<u>Step 3</u>: First or second-order analysis
Compare the chosen α_{CR} to the Eurocode 3 criterion 10, to decide whether or not it is necessary to take imperfections into account.
<u>Step 4</u>: Application of frame imperfections
To obtain the node displacements of the structure, the buckling modes are used. The one resulting from a buckling analysis corresponding to sway buckling gives the necessary information.
<u>Step 5</u>: Application of member imperfections
In the automatic procedure member imperfections need to be added to frame imperfections. The shape of the deformation is a bow; the value is the one defined by Eurocode 3 ($e_{0,d}$) and the direction is the one defined by the sway buckling of step 2.

Step 6: Non-linear analysis
Once frame and member imperfections applied, a non-linear analysis is performed. Second-order forces and moments are obtained.
Step 7: Check structure to Eurocode 3
In case of a first-order analysis, use the first-order forces and moments of step 2 to check the strength and the stability of the structure according to Eurocode 3. In case of a second-order analysis, use the second-order forces and moments of step 6 to check only the strength of the structure.

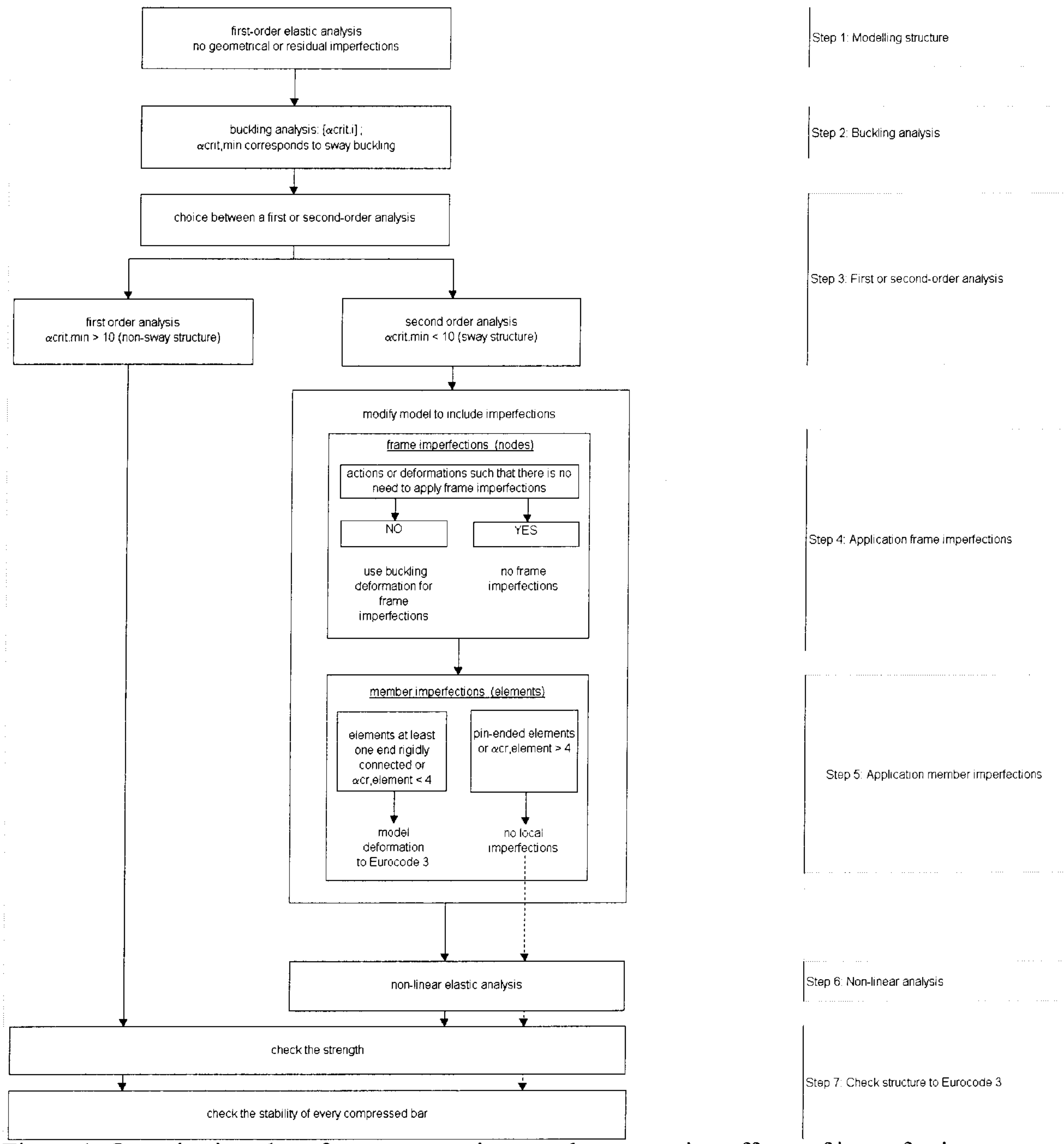


Figure 1: Organisation chart for an automatic procedure assessing effects of imperfections

KEYWORDS

Instability – Eurocode 3 – ROBOT97 – Hercule – Steel Frames – Geometrical Imperfections – Second-Order Analysis – Buckling Analysis – Automatic Procedure – Buckling shapes – Eigenvectors – Critical Buckling coefficient – Euler Buckling – Sway Frame – Non-sway Frame

NOTATIONS

Capital letters

A	cross-section
E	modulus of elasticity
F	transverse force
H	horizontal load
I	second moment of area
K_T	tangent stiffness matrix
K_0	linear stiffness matrix
K_σ	initial stress matrix
K_D	initial displacements matrix
L	system length
L_f	buckling length
N	compressive load
N_{SD}	design coefficient of the compressive force
V	vertical load
V_{CR}	elastic critical failure load
W_{el}	effective section modulus

Small letters

$e_{0,d}$	value of member imperfection
f_y	yield strength
h	height
k_c	coefficient depending on the number of columns
k_s	coefficient depending on the number of storeys
l_0	initial length
l	length

Greek letters

α imperfection factor

α_{CR} critical buckling coefficient

δ resulting displacement of the structure

ε_E nominal deformation

ε_N natural deformation

ε_G Green deformation

λ slenderness of the structure

ϕ initial sway imperfection

ϑ rotation

1. INTRODUCTION

DIMENSIONING STEEL FRAMES ACCORDING TO EUROCODE 3. PROPOSAL FOR AN AUTOMATIC PROCEDURE TO TAKE IMPERFECTIONS INTO ACCOUNT.

Structural Eurocodes are European codes, for the design of building structures, which are written by authors from several European countries. Eurocodes have been distributed for study. Each country issued National Application Documents (NAD) which are supplementary to the Eurocode and have been used experimentally in several European countries since 1992. Hope remains that in a few years there will be one set of Eurocodes applicable in all European countries.

Eurocode 3 allows the analysis of steel structures. Two methods of analysis can be distinguished to determine forces and moments arising from loading on a structure. Those methods are a first-order analysis, in which geometrical imperfections of the structure are not taken into account, and a second-order analysis, in which geometrical imperfections of the structure are taken into account. Eurocode 3 allows a possibility to do such a second-order analysis of a structure. For this type of analysis the definition of geometrical imperfections is important. But questions remain on the shape, the direction and size of the imperfections.

A second-order analysis of a structure comes down to doing a non-linear analysis of a structure. To do so, several tools are available, but the complexity of most of these implies the use of a computer. Therefore, the following structural programs were chosen: ROBOT97, Hercule and PEPmicro. Those programs allow determining, next to carrying out non-linear calculations, the critical buckling coefficient α_{CR}.

The goal of this report is to limit the problems with the application of the "imperfections of a structure", so that an automatic procedure can be proposed to take geometrical imperfections into account.

This report can be subdivided into two different parts. The first part "literature survey", compiles the necessary information taken from the literature concerning the two most important themes of this report: Eurocode 3 and non-linearity. First some clauses of Eurocode 3, concerning sway analysis and imperfections, are pointed out. Based on examples and limits to some clauses, the extent of an automatic procedure will be defined. Then the effects, causing non-linearity (qualitatively) and the necessary mathematical descriptions of non-linearity (quantitatively), are described: Green deformation and stiffness matrices. This leads to two types of analysis: linear buckling analysis and incremental analysis. Both are used in an automatic procedure. Finally, three structural analysis programs are introduced: ROBOT97, Hercule and PEPmicro. Their possibilities are briefly described. In case of an automatic procedure, only ROBOT97 and Hercule are used.

The second part comprises an "Automatic procedure to include imperfections". First, important points and limits of the literature survey will be recalled, to justify the proposed automatic procedure. Then, problematic points are studied by means of examples. These problems are the selection of the critical buckling coefficient, α_{CR}, corresponding to the relevant buckling deformation, and the way to apply frame imperfections to a structure. Finally, the automatic procedure is applied to two cases, a structural individual element and a single frame.

2. Eurocode 3: Part 1.1: General rules and rules for buildings

The goal of this project is to develop an automatic procedure to take imperfections of a structure into account. This structure will be dimensioned and checked according to Eurocode 3, [EURO3,92], using a second-order analysis.

Eurocode 3 does not always clearly state the required information on imperfections. Methods are given to define member and frame imperfections. However, Eurocode 3 remains unclear on the value of frame imperfections and on the direction to apply frame or member imperfections. Often, the suggested design methods can only be applied to specific cases. Eurocode 3 clauses on frame imperfections correspond to structures comprising 'vertical' columns and 'horizontal' beams; for structures with elements of certain angle, Eurocode 3 is difficult to apply. In case of an automatic procedure, this can cause problems.

To be able to devise a general procedure, many cases must be considered. That is why in this part, all relevant clauses in Eurocode 3 with reference to an automatic procedure will be discussed.

The research has the following limitations:
- plane structures only;
- elastic analysis;
- lateral torsional buckling will not be considered;
- elements are pin-ended or rigidly jointed.

According to Eurocode 3, the design of a structure requires four essential steps:
1. Classification of the structure as braced or unbraced;
2. Classification of the structure as sway or non-sway;
3. Assessment of the sway and non-sway imperfections;
4. Determination of forces and moments in an element by a first-order or a second-order analysis (the choice of analysis depends on the three previous points).

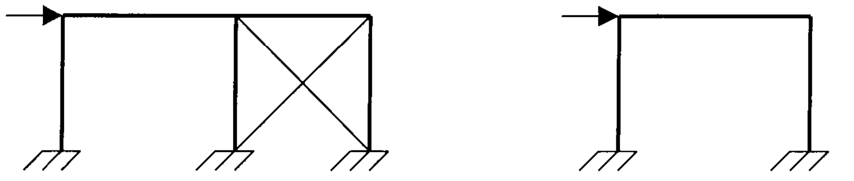
2.1 Classification of structures

Eurocode 3 regroups a large amount of clauses and information. To lead the way some criteria have been defined. Those criteria, either qualitative as quantitative, allow classification of structures. So once a structure is defined as a plane structure, the first steps, according to Eurocode 3, are to classify the structure according to bracing and sway.

2.1.1 Classification as braced or unbraced

The first classification, according to Eurocode 3, consists of finding the bracing system of a structure. Eurocode 3 states:
« *A frame may be classified as braced if its sway resistance is supplied by a bracing system with a response to in-plane horizontal loads which is sufficiently stiff for it to be acceptably accurate to assume that all horizontal loads are resisted by the bracing system.* » (art. 5.2.5.3 (1))

Any other frame shall be classified as unbraced. Figure 2.2 represents a braced system in (a) and an unbraced system in (b).



(a) Braced structure (b) Unbraced structure

Figure 2.2: Classification of bracing

The quantitative definition has been added to the previous definition:
« *A steel frame may be classified as braced if the bracing system reduces its horizontal displacements by at least 80%* ». (art. 5.2.5.3 (2))

A structure can be considered as braced if it contains a system (the bracing system) that braces the rest of the structure (the unbraced system). Examples, like in Figure 2.2, are cases where the bracing system is easily defined. Here the triangular form braces the rest of the structure. It is not the case for all structures, e.g. in Figure 2.3 the bracing system is not easily identifiable.

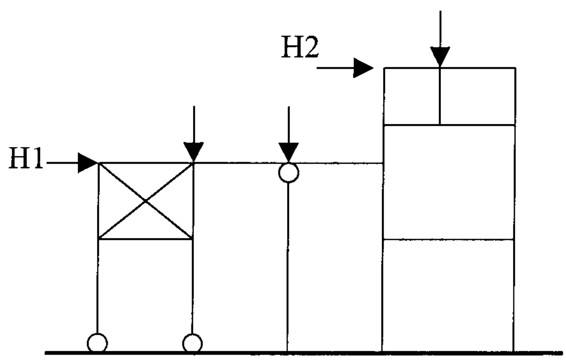


Figure 2.3: General plane frame

For the structure in Figure 2.3, it is not possible to directly indicate which part of the structure represents the bracing system. Fortunately, Eurocode 3 gives one criterion that allows isolation of the bracing system. In order to find out which part of the structure braces the rest, the following procedure can be used. First assume that a part of the structure represents the bracing system. Then, do two linear analyses of the whole structure, once with the chosen bracing system and once without, to determine the horizontal displacements. Finally, compare the results of both the analyses and check if the total horizontal displacements is reduced by at least 80%. If it is not the case, choose another part of the structure to be the bracing system, and proceed over again.

In case of an automatic procedure, the bracing system will not be considered. But the imperfections will be included, and sway stability of the structure will be checked. The bracing system, if present, is thus automatically taken into account.

2.1.2 Classification as sway or non-sway

The second classification, according to Eurocode 3, consists of finding out the stiffness resistance of the structure. Therefore the criterion used is the critical buckling coefficient. That criterion allows the analysis of the stability of structures and its classification.

Instability of structures

The determination of the critical buckling coefficient α_{CR}, consists of finding, for an arbitrary load on the structure, a load factor which will cause the structure to become unstable (very large deformations). The critical load is defined by the Euler formula:

$$V_{CR} = \frac{\pi^2 \cdot EI}{L_F^{\ 2}}$$

Where

E, modulus of elasticity

I, second moment of area

L_F, buckling length

Eurocode 3 uses this critical load to define the critical buckling coefficient α_{CR}:

$$\alpha_{CR} = \frac{V_{CR}}{V}$$

Where

V_{cr}, the elastic critical failure load in a sway mode

V, the total vertical design load

Remark: instability is often described in buckling deformations and can be subdivided into:

- non-sway buckling, one of the structural elements of the structure buckles before the rest of the structure;
- sway buckling, the critical load is such that the whole structure buckles.

Classification as sway or non-sway

The stiffness of the structure determines the classification of the structure. Eurocode 3 states:
« *A frame may be classified as non-sway if its response to in-plane forces is sufficiently stiff for it to be acceptably accurate to neglect any additional internal forces or moments arising from horizontal displacement of its nodes.* ». (art. 5.2.5.2)

Any other frame shall be classified as sway. The structure in Figure 2.4, under transverse actions, undergoes a displacement δ. The broken line represents a sway buckling.

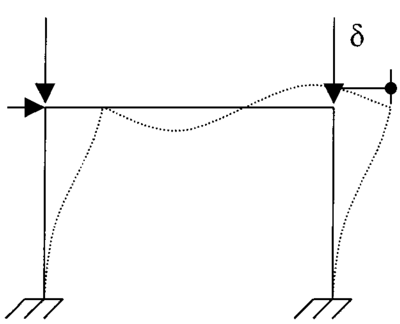


Figure 2.4: Example of a sway buckling

To define the stiffness of a structure, Eurocode 3 uses the critical buckling coefficient α_{CR}, resulting from a stability analysis of the structure. A quantitative criterion, in Eurocode 3, is defined to distinguish a sway structure from a non-sway:

- $\alpha_{CR} \geq 10$, the structure will be considered as non-sway;
- $\alpha_{CR} < 10$, the structure will be considered as sway.

This criterion 10 is associated with the fact that there is a certain relationship (α_{CR} lower than 10) between the loading and the stiffness of the structure, where the displacements cannot be neglected anymore. It requires taking displacements into account, i.e. a second-order analysis. That also means that if a second-order analysis is not necessary, there is no need to assess imperfections.

2.2 Behaviour of structures

The following choice to be made to design a structure is the type of analysis. The choice is based on the results found with the classification of structures. The classification as braced or unbraced is unimportant with respect to the automatic procedure. The classification as sway or non-sway is important for the structural behaviour.

2.2.1 The behaviour of steel structures

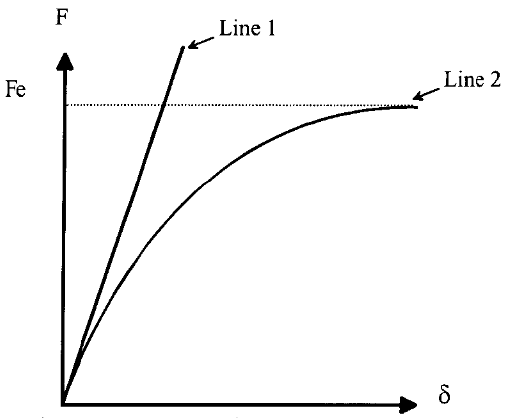
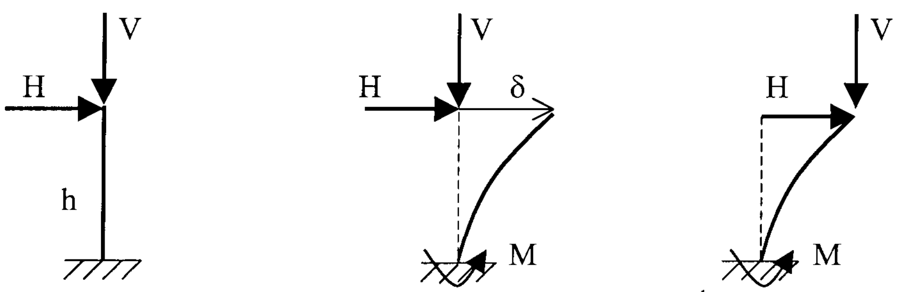


Figure 2.5: Elastic behaviour of steel structures

A restriction has been made to take into consideration only first and second-order elastic analyses. Line 1 in Figure 2.5 shows a first-order elastic analysis, whereas line 2 shows a second-order elastic analysis. Figure 2.6 shows the principal differences between these two analyses.
The second-order effects arise from the simultaneous presence of the loads and the displacements in the structure due to the application of these loads.



Theory:	1st Order	2nd Order
Displacements:	Neglected	Considered
Moments:	$M = H.h$	$M = H.h + V.\delta$

Figure 2.6: Second-order effects

2.2.2 First-order elastic analysis:

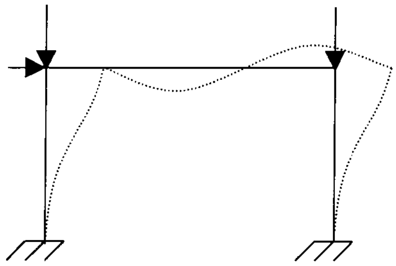


Figure 2.7: First-order deformation

The first-order elastic analysis is based on equilibrium in the undeformed state of the structure. The codes assume that the stress-strain behaviour of the material is linear. Hooke's law applies: $\sigma = E\,\varepsilon$. Figure 2.7 represents a first-order buckling shape of a single frame.

2.2.3 Second-order elastic analysis

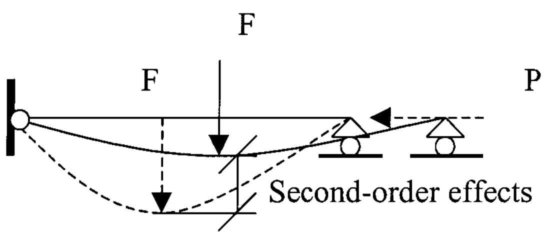


Figure 2.8: Second-order effects

Lateral deformations, due to axial forces, have to be taken into account; an example is represented in figure 2.8. These deformations can considerably modify the response of the frame. A second-order elastic analysis is obtained when formulating equilibrium for the deformed structure. Hooke's law is still applied. It is not possible to obtain directly the final state of the structure. An incremental analysis is required.

2.3 Assessment of imperfections according to Eurocode 3

The final step, according to Eurocode 3, before analysing a structure, is the assessment of imperfections. Taking into account imperfections comes down to analysing the structure non-linearly. In Figure 2.9, sway imperfections are represented by the symbol ϕ and element imperfections by the bow $e_{0,d}$. The necessary requirements will be defined for taking member and frame imperfections into account automatically in a second-order analysis.

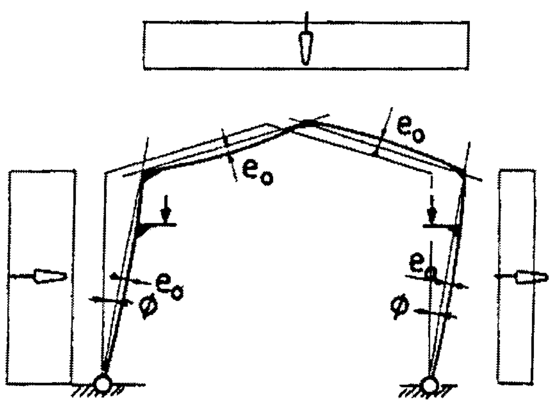


Figure 2.9: Imperfections in a structure

2.3.1 Imperfections

Taking into consideration imperfections comes down to include in the design « *the residual stresses and geometrical imperfections such as lack of verticality, lack of straightness, lack of fit and the unavoidable minor eccentricities present in practical connections.* » (art. 5.2.4.1(1))

For this particular report, not all imperfections will be considered. Only the geometrical imperfections due to lack of straightness and of verticality will be considered. Equivalent imperfections take all sorts of imperfections into account:

« *Ces imperfections géométriques équivalentes ne doivent en aucun cas être considérées comme des tolérances géométriques de fabrication et de montage à part entière. Elles prennent en compte à la fois ces tolérances ainsi que d'autres effets comme ceux des contraintes résiduelles et de l'hétérogénéité du matériau.* » (art. 5.2.4.1 (2)C) (NAD-Fr)
Translated:
« Those equivalent geometrical imperfections can in no case fully represent fabrication and erection tolerances. They take into consideration those tolerances as well as the other effects due to the residual stresses and the heterogeneity of materials. »

2.3.2 Frame imperfections and member imperfections

A frame imperfection, or sway imperfection of a structure, is considered when « *an equivalent geometric imperfection in the form of an initial sway imperfection* ϕ » is assigned to the structure. A member imperfection, or non-sway imperfection of a structure, is considered when the deformation ($e_{0,d}$) is applied, to an individual element of the structure, in the form of a bow.

In case of an automatic procedure, frame imperfections are related to the position of the joint nodes, whereas member imperfections are related to the form of the elements.

Eurocode 3 gives the possibility to replace those geometric imperfections by equivalent forces; their values approximate the effects of the different imperfections.

« *Imperfections shall be allowed for in the analysis by including the appropriate additional quantities, comprising frame imperfections, member imperfections and imperfections for analysis of bracing systems.* » (art. 5.2.4.2(1))

« *On entend par quantités additionnelles, soit des imperfections affectant la géométrie de l'ossature, soit des charges simulant des effets équivalents à ces imperfections. La nature et l'amplification de ces quantités additionnelles peuvent dépendre du type d'analyse globale effectuée* » (art. 5.2.4.2 (1)C) (NAD-Fr)
Translated
« Additional quantities are imperfections applied either by influencing the geometry of the structure or by applying loads that simulate the equivalent effects of those imperfections. The sort and amplification of those additional quantities can depend on the type of sway analysis made »

Specific cases are described in Eurocode 3:
« *Les effets des imperfections globales de l'ossature peuvent être généralement négligés dans les combinaisons d'actions où interviennent des charges horizontales significatives telles que celles résultant de l'action du vent.* » (art. 5.2.4.2(2)A) (NAD-Fr)
Translated

« The effects of frame imperfections can generally be neglected for combined actions where a significant horizontal load is present, such as the one resulting from the wind »
In case of the presence of a horizontal force, it might not be necessary to apply frame imperfections.

Eurocode 3 proposes an alternative criterion, than the stiffness, to fall back on a first-order analysis.
« *The effects of member imperfections (see 5.2.4.5) may be neglected when carrying out the global analysis of frame, except in sway frames (see 5.2.5.2), in the case of members, which are subjected to axial compression, which have moment-resisting connections, and in which:*

$$\bar{\lambda} > 0.5 \cdot \sqrt{\frac{A \cdot f_y}{N_{SD}}}$$

Where
N_{SD}, the design coefficient of the compressive force
$\bar{\lambda}$, the in-plane non-dimensional slenderness (see 5.5.1.2), calculated using a buckling length equal to the system length. » (art. 5.2.4.2(4))

The following comment has been made on that article:
« *La présence simultanée dans un élément d'une compression importante et d'une imperfection initiale en forme d'arc génère aux extrémités de celui-ci des effets qui peuvent modifier de façon sensible le comportement global d'une ossature. On considère que ceci intervient pour tout élément comprimé à plus du quart de son effort axial critique élastique N_{CR}, calculé pour la barre bi-articulée* $N > \frac{N_{CR}}{4}$
Dans le cadre de l'analyse globale, qui doit alors être au second ordre, les imperfections peuvent n'être introduites que pour les éléments remplissant la condition de ci-dessus.

Il est de bonne pratique d'effectuer dans un premier temps l'analyse globale sans tenir compte de ces imperfections et de vérifier a posteriori si l'un quelconque des éléments remplit la condition énoncée. Dans les cas, en général peu nombreux, où la réponse est affirmative, l'analyse globale doit alors être reprise pour les combinaisons d'actions concernées » (art. 5.2.4.2(4)C) (NAD-Fr)
Translated
« In an element simultaneous presence of an important compression and an initial bow imperfection generates at the end nodes effects that can modify in a sensible way the sway behaviour of a structure. This happens for all elements compressed over the quarter of their elastic critical axial load N_{CR}, determined for the pin-ended element $N > \frac{N_{CR}}{4}$
For the global analysis, that then has to be in a second-order, the imperfections can only be applied for the elements answering the above criteria.

It is usually done to first do a global analysis without considering the imperfections and to verify afterwards if any of the elements answers the above criterion. In the cases, mostly not numerous, where the response is affirmative, the global analysis has to be redone for the concerned actions combinations »

Note that the two following criteria are equivalent:

$$\bar{\lambda} > 0.5 \cdot \sqrt{\frac{A \cdot f_y}{N_{SD}}} \quad \equiv \quad N_{sd} > \frac{N_{CR}}{4}$$

For an automatic procedure, that remark is essential. Because it proposes, in case of a second-order analysis, to apply imperfections only to members where $\alpha_{CR} < 4$.

Next to this criterion, for an automatic procedure, pin-ended elements can as well be analysed separately. For the automatic procedure, the buckling shape is important. Pin-ended elements can only be of influence in case of compression, because the joints do not transmit any moments to the structure. Therefore, there is no obligation to apply member imperfections, but those elements are checked later in the procedure.

So, in the case of an automatic procedure, it is not necessary to apply member imperfections for:
- pin-ended elements;
- elements for which: $\alpha_{CR} > 4$.

2.3.3 Approximation frame and member imperfections

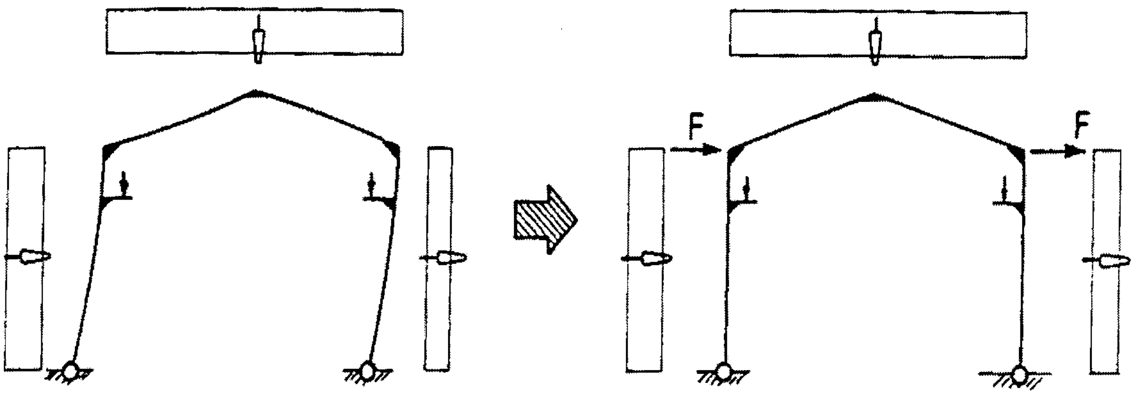
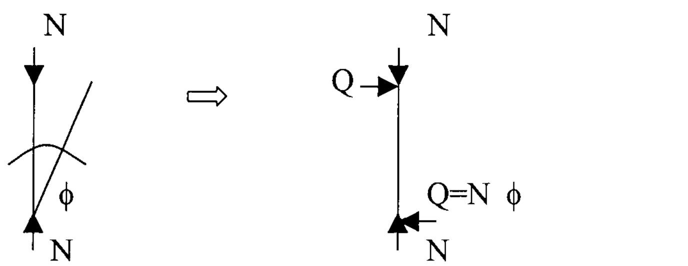


Figure 2.10: Assessment of frame imperfections as equivalent forces

Frame imperfections
The assessment of frame imperfections and the modification as equivalent forces are represented in Figure 2.10. The deformation of vertical elements causes a buckling moment, represented by a couple of forces Q, in Figure 2.11.
So that:
$Q = N \cdot \phi$

Where
« *$\phi = k_c * k_s * \phi_0$.*
$\phi_0 = 1/200$
k_c depending on the number of columns per plane
k_s depending on the number of storeys. » (art. 5.2.4.3(1))



Geometric imperfection Geometric equivalent forces

Figure 2.11: Frame imperfections

« *These initial sway imperfections apply in all horizontal directions, but need only be considered in one direction at a time* » (art. 5.2.4.3(4))
« *If more convenient, the initial sway imperfection may be replaced by a closed system of equivalent horizontal forces* » (art. 5.2.4.3(6))
Those definitions are only valid for specific cases where the structure is defined with "horizontal" / "vertical" elements. For each horizontal displacement, a system of actions has to be applied at the end nodes of the elements. This is to prevent the introduction of any additional actions.

« *The horizontal reactions at each support should be determined using the initial sway imperfection and not the equivalent horizontal forces. In the absence of actual horizontal loads, the net horizontal reaction is zero* » (art.5.2.4.3(8)) [EURO]

For an automatic procedure, if it is necessary to apply frame imperfections, then it must be verified that no additional actions are introduced; that could disturb global equilibrium.

For general cases, the terms like "horizontal", "vertical", "columns", "beams" and "storeys" are difficult to interpret. Examples are represented in Figure 2.12.

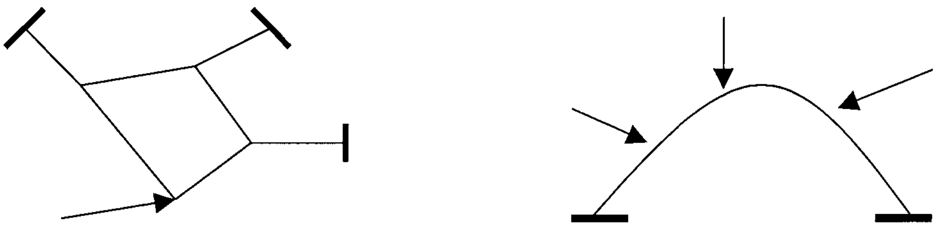


Figure 2.12: General structural forms

In those cases it is impossible to apply frame imperfections following Eurocode 3. Frame imperfections only can be assigned to structures by applying a certain displacement vector $\vec{u}$ to different nodes. Therefore, it is necessary to suggest a general way to take imperfections into account.

Determination of the member imperfections
a) Application of member imperfections:

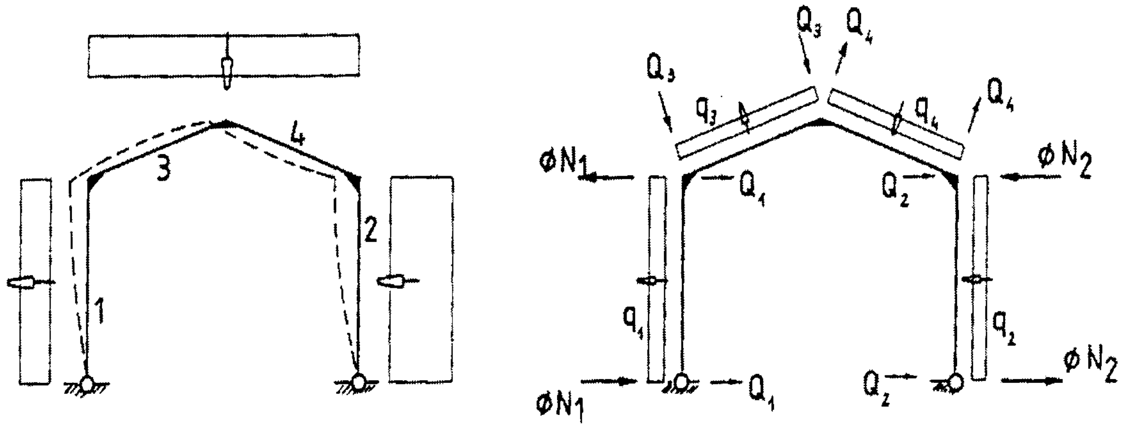


Figure 2.13: Equivalent member forces

The way to generate member imperfections, as represented in Figure 2.13, is not clearly defined in Eurocode 3. Eurocode 3 gives formulas for the initial deflection $e_{0,d}$, but does not define the shape

of the bow. The question arises whether the form of that bow can be a circle, a sinusoid or a parabola.
« *Second-order analysis of a member shall incorporate the appropriate equivalent initial bow imperfection corresponding to the relevant buckling curve, depending on the method of analysis and type of cross-section verification* » (art. 5.5.1.3(4))

To define $e_{0,d}$, it is necessary to consider the buckling length. « *Normally the effects of imperfections on members design shall be incorporated by using the appropriate buckling formulae given in this Eurocode* » (art. 5.2.4.5(1))

« *Alternatively, for a compression member, the initial bow imperfection specified in 5.5.1.3 may be included in a second-order analysis of the member* » (art. 5.2.4.5(2))

« *Where it is necessary (according to 5.2.4.2) to allow for member imperfections in the global analysis, the imperfections specified in 5.5.1.3 shall be included and second-order global analysis shall be used* » (art. 5.2.4.5(3))

b) Buckling length and member imperfections

« *The buckling length l of a compression member with both ends effectively held in position laterally, may conservatively be taken as equal to its system length L* » (art. 5.5.1.5(1))

« *Le maintient latéral peut:*
- *Soit être réel;*
- *Soit résulter d'une hypothèse adoptée dans un modèle de calcul.* » (art. 5.5.1.5(1)C)

(NAD-Fr)
Translated
« The holding element can be:
- real;
- resulting from a calculation model. »

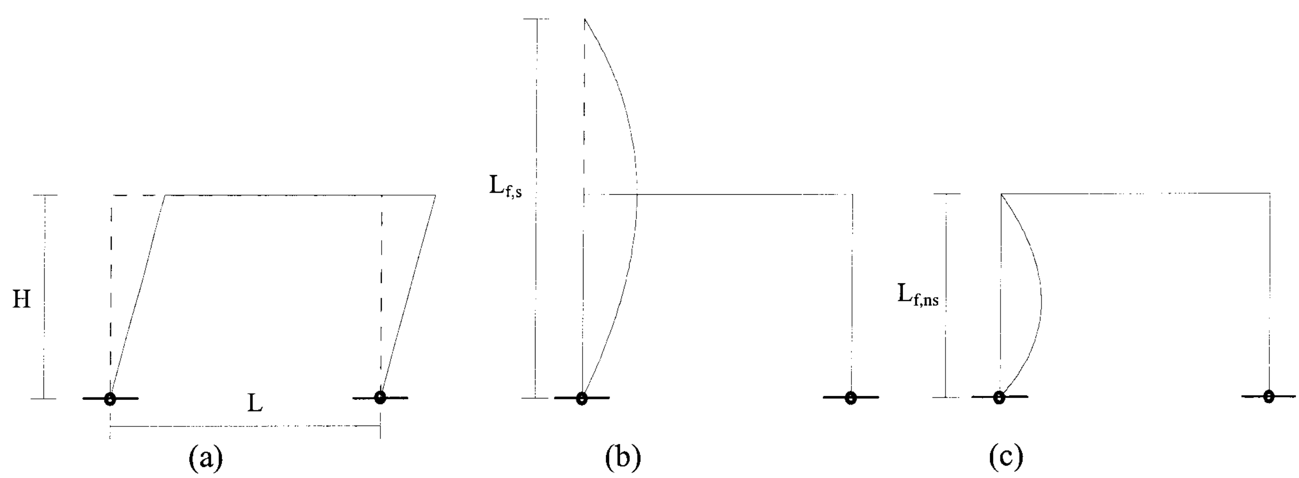


Figure 2.14: Application of frame imperfection (a), buckling length of a sway structure (b) and of a non-sway structure (c)

The automatic procedure suggests applying frame and member imperfections to structures. Figure 2.14 (a) represents the application of frame imperfections to a single frame. In case of an automatic application, frame imperfections are obtained by doing a linear buckling analysis of the structure (see examples from chapter 7 for more details). Member imperfections are applied afterwards. According to Eurocode 3, to define the value of member imperfections it is necessary to know the buckling length. The buckling length of a structure is the length of the half sinus. In case of an automatic application, the resulting buckling length from the structural programs will be the one corresponding to a sway structure (see Figure 2.14 (b)). The buckling length is longer than the system length, which also means that $e_{0,d}$ can be very large. For member imperfections, the buckling length is two long. Eurocode 3 allows to consider the deformed structure (by frame imperfections) as non-sway and set the buckling length equal to the system length of each element, as in Figure 2.14 (c).
In case of an automatic procedure, applying imperfections comes down to adding the imperfections of Figure 2.14 (a) and Figure 2.14 (c).

c) Definition of member imperfections

Value of member imperfection
Eurocode 3 defines the imperfection $e_{0,d}$:

$$e_{0,d} = \alpha \cdot \left(\bar{\lambda} - 0.2\right) \cdot k_{\gamma} \cdot \frac{W_{el}}{A}$$

Where
α, imperfection factor
λ, slenderness
k_{γ}, coefficient depending on the material and the slenderness
W_{el}, effective section modulus
A, cross-section

Shape of the imperfection
Eurocode 3 describes the shape of the imperfection as a bow. The exact form of that bow is not defined in Eurocode 3. An analysis of several cases showed that resulting forces and moments, after applying imperfections following a sinusoid, a parabola or a circle arc, could be considered as equal.

Direction of the imperfection
The automatic procedure suggests adding member imperfections to frame imperfections. The only way to apply imperfections is to add intermediate nodes to the elements. Member imperfections have to be applied on the unfavourable side. Refer to the procedure in chapters 5.2 and 5.4, and to the examples of chapter 7, for more explanations.

d) Eurocode 3 and the second-order analysis

The following clauses give the necessary information concerning second-order analysis of the structure.

« *Second-order analysis of a member shall incorporate the appropriate equivalent initial bow imperfection corresponding to the relevant buckling curve, depending on the method of analysis and type of cross-section verification* » (art. 5.5.1.3(4))

« *The equivalent initial bow imperfection shall also be used where it is necessary (according to 5.2.4.5) to include member imperfections in the global analysis* » (art. 5.5.1.3(5))

« *When the imperfections are used, the resistance of the cross-sections shall be verified in 5.4, but using* γ_{M1} *in place of* γ_{M0}. » (art 5.5.1.3(6))

« *Dans le cas d'un élément non uniforme, ou comme alternative dans le cas d'un élément uniforme, la résistance de l'élément comprimé et fléchi sans risque de déversement peut être vérifiée en opérant une analyse locale au second ordre sur l'élément isolé, en présence de l'imperfection initiale en arc. Cette analyse au second ordre de l'élément isolé peut être faite dans les conditions précisées ci-après. Telle qu'elle est exposée, cette analyse ne prend en aucun cas en compte le risque éventuel de déversement qui est à vérifier avec le critère adéquat de 5.5.4.*

- *Analyse locale dans un plan principal d'inertie*

L'analyse locale au second ordre à opérer sur un élément comprimé et fléchi pour vérifier sa résistance au flambement dans le plan considéré peut être réalisé dans les conditions suivantes :

- *L'élément est considéré avec sa longueur d'épure L, en présence de l'imperfection initiale en arc d'amplitude* $e_{0,d}$ *(centrée sur cette longueur d'épure) calculée sur la base de la longueur de flambement Lf de l'élément, Lf correspondant au mode d'instabilité à nœuds fixes dans le plan considéré. L'imperfection* $e_{0,d}$ *doit être introduite dans le sens le plus défavorable eu égard à celui des efforts de liaison et des charges éventuellement appliquées sur l'élément dans le plan considéré;*
- *L'élément est supposé articulé à ses deux extrémités, elles-mêmes supposées maintenues latéralement. (nœuds fixes);*
- *L'élément est soumis :*
  . à ses extrémités, aux efforts de liaison (sollicitations) déterminés par l'analyse globale de l'ossature.
  . aux charges éventuellement appliquées entre ses extrémités dans le plan considéré.

Cette analyse permet de déterminer, dans le plan considéré, le moment fléchissant amplifié (M+ΔM) en chaque section de l'élément en introduisant ainsi le moment fléchissant supplémentaire ΔM dû aux effets du second ordre locaux en présence de l'imperfection équivalente initiale $e_{0,d}$.
... » (art. 5.5.4.(1)A) (NAD-Fr)
Translated
« For a non-uniform element, or as an alternative for a uniform element, the resistance of a member with combined axial force and moment and no lateral torsional buckling can be checked with a second-order non-sway analysis on the isolated element, which has the appropriate equivalent initial bow imperfection. This second-order non-sway analysis of the isolated element can be done as follows. As described here, this analysis does not take into account the lateral torsional buckling that has to be check with the appropriate criteria of 5.5.4.

- In-plane non-sway analysis

The non-sway second-order analysis to be used on a member with combined axial force and moments to check the in-plane buckling resistance can be realised by the following points:

- the element is considered with its system length L, with the appropriate equivalent initial bow imperfection of deflection $e_{0,d}$ (centred on the system length) calculated on the base of the buckling length L_f of the element, L_f of a non-sway mode. The imperfection $e_{0,d}$ must be introduced in the unfavourable direction, regarding the joint forces and moments and eventual loads applied to the element;
- the element is supposed to be pin-ended, the extremities are supposed to be held. (non-sway);

- the element is submitted to:
  - at the end nodes, joint forces and moments (loading) determined by a global analysis of the structure;
  - eventual loads applied at the end nodes.

This analysis allows to determine, in the plane of the structure, the second-order moment (M+ΔM) in each cross-section of the element thus introducing an additional buckling moment ΔM due to the non-sway second-order effects caused by the appropriate equivalent initial bow imperfection of deflection $e_{0,d}$ »

Now the cross-section strength of the members can be verified with the second-order buckling moment.

Eurocode 3 proposes alternative methods to approach a second-order analysis. Though it is not the goal to "avoid" a second-order analysis, it is interesting to note that there is a possibility to do so.

« *When elastic global analysis is used, the second-order effects in the sway mode shall be included, either directly by using second-order elastic analysis, or indirectly by using one of the following alternatives:*

(a) First-order elastic analysis, with amplified sway moments,

(b) First-order elastic analysis, with sway mode buckling lengths.

» (art. 5.2.6.2(1))

« *In the amplified sway moments method, the sway moments found by a first-order elastic analysis should be increased by multiplying them by the ratio:*

$$\frac{1}{1-\frac{1}{\alpha_{CR}}}$$

Where α_{CR} is the critical amplification coefficient » (art. 5.2.6.2(3))

« *The amplified sway moments method should not be used when the elastic critical load ratio V_{SD}/V_{CR} is more than 0.25* » (art. 5.2.6.2(4))

« *Sway moments are those associated with the horizontal translation of the top of a storey relative to the bottom of that storey. They arise from horizontal loading and may also arise from vertical loading if either the structure or the loading is asymmetrical.* » (art. 5.2.6.2(5))

« *When the amplified sway moments method is used, in-plane buckling lengths for the non-sway mode may be used for member design.* » (Art. 5.2.6.2(7))

« *When first-order elastic analysis, with sway-mode in-plane buckling lengths, is used for column design, the sway moments in the beams and the beam to column connections should be amplified by at least 1.2 unless a smaller coefficient is shown to be adequate by analysis.* » (art 5.2.6.2(8))

2.4 Summary chart and table of Eurocode 3

The main points described in the previous part can be represented in the summary chart of figure 2.15. This chart leads to the type of analysis that needs to be applied, as described in Table 2.1.

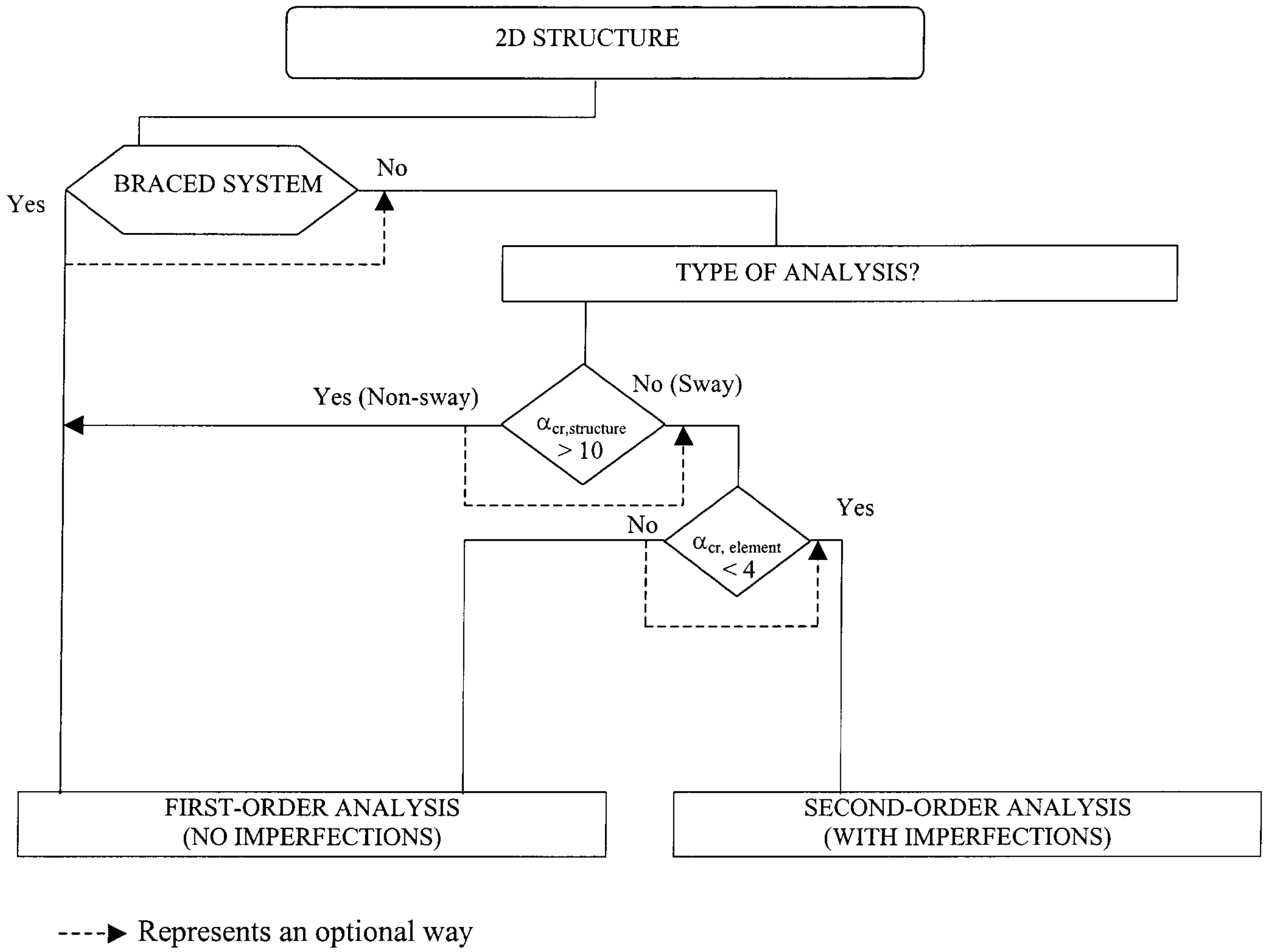


Figure 2.15: Organisation chart of Eurocode 3

Table 2.1: Classification of the structure and type of analysis

Critical buckling coefficient:	The structure is:	The analysis of the structure:
$\alpha_{cr,structure} > 10$	Non-sway	First-order
$\alpha_{cr,structure} < 10$ and $\alpha_{cr,element} > 4$	Sway	First-order Second-order
$\alpha_{cr,structure} < 10$ and $\alpha_{cr,element} < 4$	"Very" Sway	Second-order

3. Non Linearity in a structure

In order to implement an automatic procedure, some of the mathematical parameters have to be described. The structure has been classified according to Eurocode 3; the bracing classification is neglected, while the critical buckling coefficient allows the classification as sway or non-sway. A second-order analysis implies to take imperfections into account. The second-order analysis is also called a non-linear analysis of structures. First the geometrical non-linearity will be described, followed by the important clauses for an automatic procedure, the deformation and the matrices. This leads to the different numerical tools used for a linear and non-linear analysis. Finally, several computer programs based on the theory are introduced.

3.1 Geometrical non-linearity

The automatic procedure, proposed for this report, is based on the automatic application of the geometrical imperfections necessary for a non-linear analysis of structures. The effects that cause non-linearity are now described.

3.1.1 Effects that cause geometrical non linearity

In non-linear cases, equilibrium is formulated in the deformed state. The lateral stiffness of the structure determines the displacements. External loads cause those displacements; they can have several implications for the structure.

Three different cases of geometrical non-linearity can change the stiffness of the structure:

1) Effects due to large deformations:
   The large displacements, of certain nodes compared to others of the same structure, cause a modification of the stiffness.
2) Effects due to large rotations:
   The large rotation of elements changes the stiffness and implies a modification of the global structure.
3) Effects due to the stress-stiffening:
   An axial compression (or tension) force contributes to a relaxation (or stiffening) of the element.

3.1.2 Factor for non-linearity: Deformations

After a load has been applied, the structure will have a certain tendency to deform (depending on its rigidity). The deformation is the important factor for any analysis of the structure. The literature distinguishes three types of deformation: the nominal deformation, also called «engineering deformation» ε_E, the natural deformation ε_N and the Green - Lagrange deformation ε_G

1) The nominal deformation
An axial force F is applied to an element, as represented in Figure 3.1; the displacement is Δl_0.

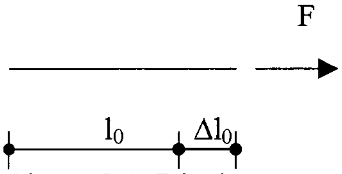


Figure 3.1: Displacement due to the force F

The nominal deformation is defined as:

$$\varepsilon_E = \frac{l - l_0}{l_0} = \frac{\Delta l_0}{l_0}$$

2) The natural deformation
The natural deformation is defined as:

$$\varepsilon_N = Ln\left(\frac{l}{l_0}\right) = Ln\left(1 + \frac{\Delta l_0}{l_0}\right)$$

Δl_0 is often small, after a first-order development of that deformation the engineer deformation can be obtained.

3) Green deformation
The Green deformation is the one considered in this report; it is defined for a point A of the cross-section as represented in Figure 3.3, and is defined:

$$\varepsilon_G = \frac{du}{dx} - y \cdot \frac{d^2v}{dx^2} + \frac{1}{2}\left[\left(\frac{dv}{dx}\right)^2 + \left(\frac{du}{dx}\right)^2\right]$$

Figure 3.2 represents an element of length dx that after loading becomes ds. The Green deformation uses the vertical and horizontal deformations du and dv. The Green deformation here above is for an unknown point A of the cross-section of an element. This report uses a reduced Green deformation formula, for a point G on the neutral axis:

$$\varepsilon_{G,X} = \frac{du}{dx} + \frac{1}{2}\left[\left(\frac{dv}{dx}\right)^2 + \left(\frac{du}{dx}\right)^2\right]$$

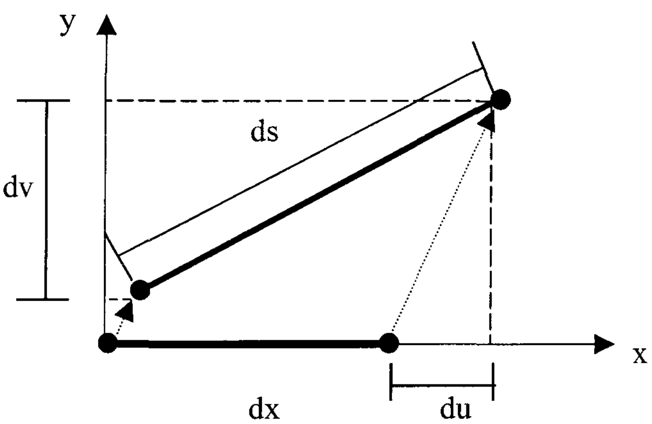


Figure 3.2: Deformation of an element

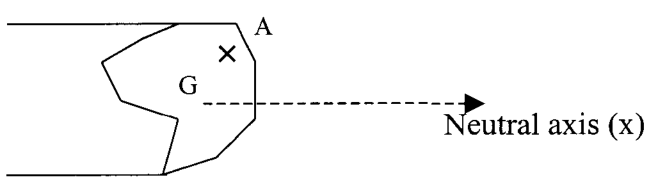


Figure 3.3: Cross-section of the element

3.2 The Green deformations and the stiffness matrices

Non-linear effects are described in a practical way. A general mathematical approach is used to understand the different programs. The so-called Green deformations and the stiffness matrices are introduced. Annexes C and D describe further in detail those points.

3.2.1 The Green deformations: Formulation

The Green deformation is defined as, [GALAM,76]:

$$\varepsilon_{G,X} = \frac{du}{dx} + \frac{1}{2}\left[\left(\frac{dv}{dx}\right)^2 + \left(\frac{du}{dx}\right)^2\right]$$

The first term $\frac{du}{dx}$ corresponds to the basic linear relation, without instability effects.

The second term $\left(\frac{dv}{dx}\right)^2$ represents the axial deformation due to the elongation of the neutral axis in deformed stated.

The last term $\left(\frac{du}{dx}\right)^2$ can be considered as proportional to the square of the axial deformation and be neglected for small deformations elastic analyses.

3.2.2 The Green deformations: Specific cases

The literature distinguishes three cases for the Green deformations. Those cases also represent the three types of calculations that the several programs use.

1) Linear case

The rotations and displacements are low:

$$\frac{du}{dx} \leq 1$$

$$\frac{dv}{dx} \leq 1$$

The terms of order two can then be neglected; the classical formula is obtained:

$$\varepsilon_{G,X} \approx \frac{du}{dx}$$

For this report the method is called: linear analysis.

In France, the Netherlands and Germany this case is used for the « first-order analysis » of a structure.

2) Non-linear case - Large rotations – Small deformations

The axial deformation remains small compared to the rotation terms:

$$\frac{du}{dx} < \frac{dv}{dx}$$

$$\frac{du}{dx} \leq 1$$

The deformation becomes:

$$\varepsilon_{G,X} \approx \frac{du}{dx} + \frac{1}{2}\left(\frac{dv}{dx}\right)^2$$

This case is frequently used in non-linear analyses of structures. Because it takes into consideration the large displacements caused by the rotation of the system and it includes the small deformations.

For this report the method is called: non-linear analysis with small deformation.
In France, the Netherlands and Germany this case is used for the « second-order analysis » of a structure.

3) Non-linear case – Large rotations – Large deformations

The deformations $\frac{du}{dx}$ and the rotations $\frac{dv}{dx}$ are important and cannot be neglected anymore. All the terms of the Green deformation have to be taken into account:

$$\varepsilon_{G,X} = \frac{du}{dx} + \frac{1}{2}\left[\left(\frac{dv}{dx}\right)^2 + \left(\frac{du}{dx}\right)^2\right]$$

For this report the method is called: non-linear analysis with large deformation.
In France and the Netherlands this case is used for the « second-order analysis, large deformations » of a structure. Whereas in Germany this case is used for the « third-order analysis » of a structure.

3.2.3 The Green deformations: Graphical representation

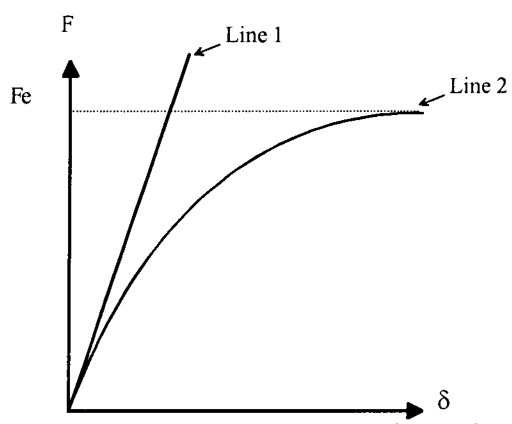


Figure 3.4: Graph of the Green Deformation

Consider the Green deformation for a point G on the neutral axis of an element:

$$\varepsilon_G = \frac{du}{dx} + \frac{1}{2}\left[\left(\frac{dv}{dx}\right)^2 + \left(\frac{du}{dx}\right)^2\right]$$

Figure 3.4 represents:

- line 1: linear part of the Green deformation

$$\varepsilon_G = \frac{du}{dx}$$

- line 2: the whole deformation

$$\varepsilon_G = \frac{du}{dx} + \frac{1}{2}\left[\left(\frac{dv}{dx}\right)^2 + \left(\frac{du}{dx}\right)^2\right]$$

3.2.4 The matrices

The Green deformations are developed in the annexes to understand the way the different programs, used for this report, work. The programs ROBOT, Hercule and PEPmicro are all based on matrix calculations. To define those matrices, the Green deformations are considered.

« *For a non-linear analysis, it is recommended to do a study based on virtual displacements. That corresponds to a potential energy method approach.* » (Thomas, 1970)

Important to know is that the different structural programs are based on matrix calculations. In case of analyses of structures, the so-called tangent stiffness matrix is used; it contains three different terms:

$[K_T] = [K_0] + [K_\sigma] + [K_D]$

- K_0, initial stiffness matrix
- K_σ, initial strain matrix
- K_D, large displacements matrix

The matrix K_0 depends on the stiffness of the element, K_σ on compression actions applied to the element whereas K_D comprises second-order terms.

In case of a linear analysis of structures, the matrix $[K_T]$ will only contain the matrices $[K_0]$ and $[K_\sigma]$. And for non-linear analyses, the three different matrices are used.

3.3 Numerical tools to solve linear and non-linear systems

The literature distinguishes two methods: A "determinant" method, called linear analysis or first-order analysis and an incremental method, called non-linear analysis or second-order large / small displacements method.

Before analysing the solving methods, some elementary situations that can arise in structural instability are described, especially the different behaviours that a structure undergoes after instability. [GALAM,76]

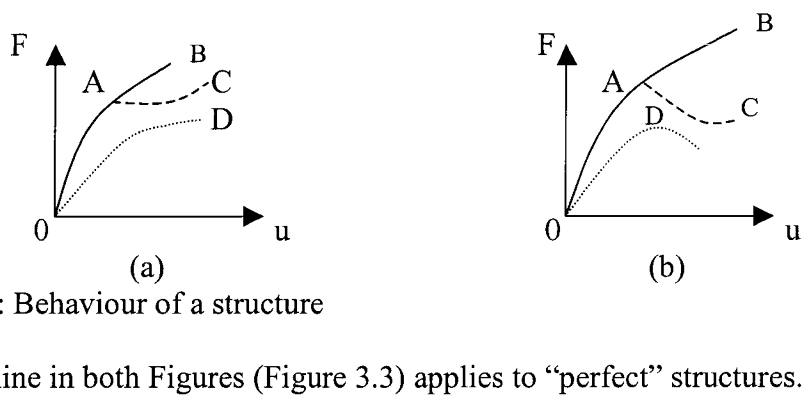


Figure 3.3: Behaviour of a structure

The solid line in both Figures (Figure 3.3) applies to "perfect" structures. In these, the structure first displaces along the "fundamental path" (OAB), with bifurcation to a secondary path at point A. the post buckling path AC may rise (Figure 3.3a) or descend (Figure 3.3b), depending on the characteristics of the structure and the loading.

For certain structural forms, or when fabrication imperfections are taken into account for the types of perfect structures that are portrayed with the solid lines, the load displacements behaviour follows the paths indicated by the dashed lines, OD. Structures with a rising postbuckling path

(Figure 3.3a) will have a strength exceeding the bifurcation load. The load-carrying capacity of a real structure with a descending postbuckling path (Figure 3.3b) will be less than the bifurcation load of the corresponding perfect structure unless the load-displacement path rises again at larger displacements. This maximum load-carrying capacity, or limit point, occurs at point D.

Then for finite-element analysis, at least four different circumstances are needed:
- a general non-linear analysis for tracing of the load displacement path;
- calculation of the bifurcation point;
- determination of the load-displacement response along a postbuckling path;
- calculation of limit points.

This report considers the two first points. First the method to determine the bifurcation point is described; it is followed by the possibilities to approach the non-linear behaviour of the structure.

3.3.1 Linear buckling analysis, bifurcation analysis

For many structures, the prebuckling load-displacement path is considered as linear. This method is used to define the load corresponding to the instability of the structure. Then the different critical buckling coefficient of the different buckling modes can be determined.

There is a certain force at which the structure buckles. For a linear case, it comes down to a deformation of the structure.

$$\lambda F_X = EA\frac{du}{dx}$$

With λ a scalar that defines the intensity of the load F_X. F_X is obtained in an analysis for a reference intensity of the applied loads.

After applying the principle of stationary potential energy, the following buckling criterion is obtained:

$$[K_0 + \lambda \cdot K_1] \cdot \{\delta\Delta\} = 0$$

or

$$[K_0 + \lambda \cdot K_1] = 0$$

The coefficient λ is a scalar, in the mathematical literature it is called eigenvalues. The solution of the criteria above gives the several eigenvalues corresponding to the different solutions. The critical buckling coefficient (Euler criterion) corresponds to the smaller positive coefficient.

The critical buckling coefficient, as explain before, is not only used to determine the bifurcation point of the structure. It is also used in the automatic procedure to classify the structure. That criterion will be used to define the necessity to do a second-order analysis of the structure and the necessity to apply imperfections.

3.3.2 Non-linear analysis, incremental techniques

[FREY,78] described the different principal techniques for the non-linear analysis. A non-linear analysis of the structure comes down to approximating the non-linear behaviour of the structure. This can either be done in a direct way or in an incremental way.

Different methods are used to approximate the non-linear behaviour of a structure. Those methods are incremental. Frey's studies showed that only four methods are interesting and currently used:

1) « step by step » method;
2) « step by step » method with residue;
3) Newton-Raphson method;
4) Newton-Raphson method with residue.

Annex E shows the graphical representation of the different tools.

3.3.3 The different programs for this report, and their possibilities of application

The different tools and theoretical information for this report are described in the previous part and in the annexes. The Green deformation forms the basis to determine the different matrices so that the different numerical tools can solve the linear or non-linear problem.

For this report three programs were chosen, [ROBOT,96] , [HERCU,98] and [PEP,95]:

- ROBOT 97, version 12.5;
- HERCULE, version 33.01;
- PEPmicro, version 2.

The chart of Figure 3.4 represents the different possibilities for practical applications:

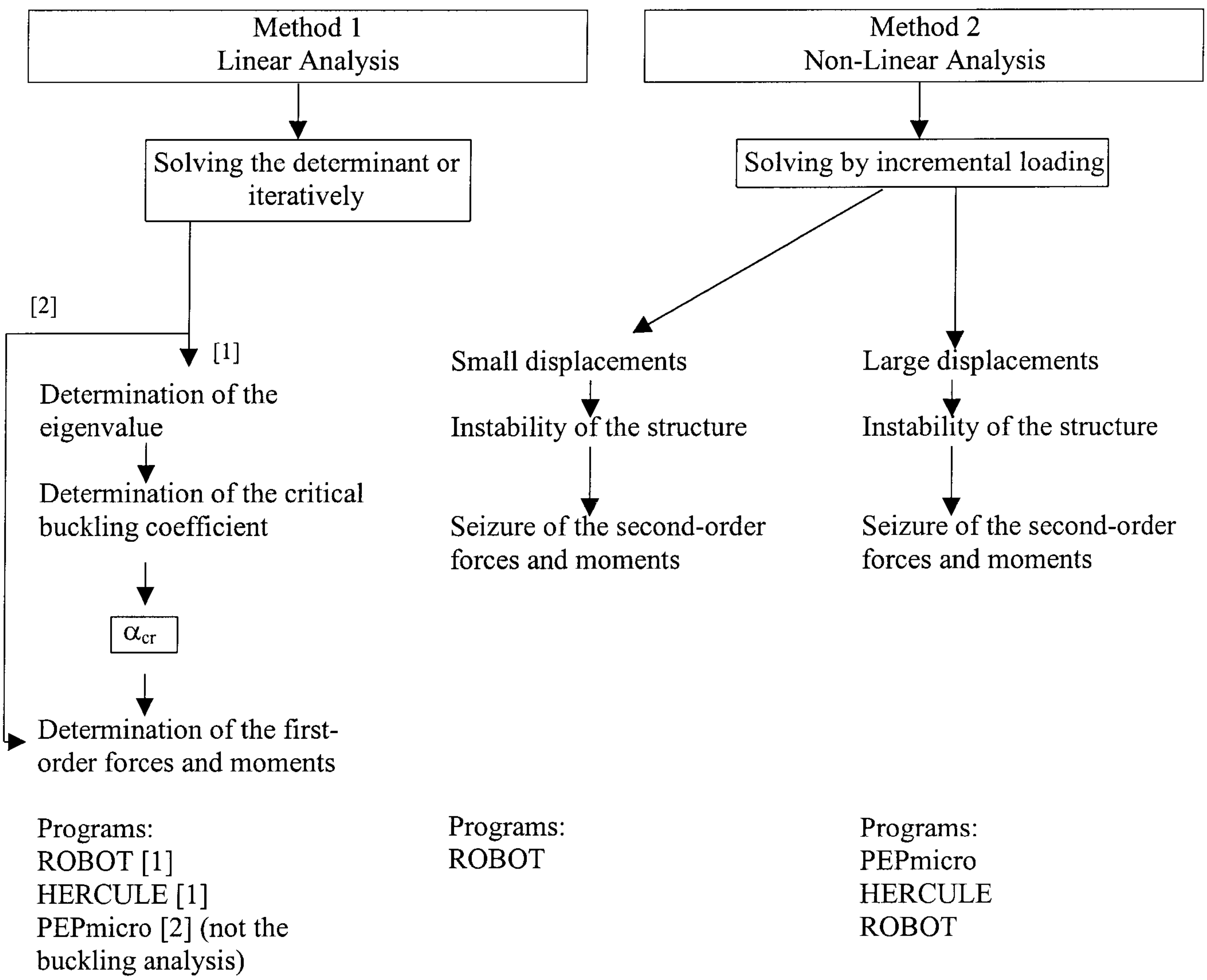


Figure 3.4: Possibilities of the different programs

4. Summary of the literature survey

For the automatic procedure, the way to apply member and frame imperfections on a structure is important. But in general cases Eurocode 3 seems to have limits.

The following points need further research:

- in Eurocode 3, structures are classified as braced or unbraced; in the automatic procedure this will not be done. The braced and unbraced structures are treated equally;
- Eurocode 3 only considers structures with horizontal beams and vertical columns. In general cases, for frame imperfections, this may cause problems if members have an arbitrary angle. Another way to apply frame imperfections will therefore be suggested;
- determining the value of the maximum frame imperfection deformation is for general cases not defined; once more a method needs to be suggested;
- elements that are pin-ended or for which $\alpha_{CR} > 4$, can be treated separately. They are either considered later in the procedure, and then stability checks of these elements need to be done. Or, for the automatic procedure, they are taken into account in the global analysis of the structure.

The conclusions are for the theoretical part:

- it is necessary to classify the structure as sway or non-sway by using the critical buckling coefficient α_{CR}. This allows the justification of the use of a non-linear analysis for a sway structure;
- the buckling length of an element in a structure, for the application of member imperfections in the automatic procedure, equals the system length;
- the Green deformation is used in all three programs. The principle of stationary potential energy allows the determination of the stiffness matrices;
- taking the chance to simplify the Green deformation allows three types of analysis:
  - a linear analysis;
  - a non-linear analysis with small displacement;
  - a non-linear analysis with large displacement.

Three structural programs were selected:

- ROBOT includes all three types of analyses. It can also determine the critical buckling coefficient;
- Hercule only allows linear and non-linear large displacement analyses. One of its modules calculates the critical buckling coefficient;
- PEPmicro does the same analyses as Hercule, but it can only be used to determine the forces and moments in the elements.

Knowing that PEPmicro does not determine the critical buckling coefficient, the automatic procedure will be considered for the programs ROBOT and Hercule.

Based on these aspects an automatic procedure can now be proposed. Then some of the most important steps will be described more in detail to sharpen the procedure. Finally, based on two examples, the procedure will be verified in the next section of the report.

5. Main steps of the procedure

When dimensioning a structure or checking its individual members according to Eurocode 3, two main steps can be distinguished:
- classification of the structure (sway / non-sway);
- assessment of the imperfections (frame and member).

The classification can be done with the critical buckling coefficient α_{CR}. The value of the critical buckling coefficient will be compared to the criterion 10; the structure will be classified as sway for $\alpha_{CR}<10$ or non-sway otherwise.

For non-sway structures, Eurocode 3 suggests a first-order analysis of the structure. The structure is considered as theoretically perfect; the elements are to be checked on static strength and stability. For a sway structure, Eurocode 3 offers the possibility to do a second-order analysis with imperfections taken into account. Then only a static strength check of the structure is necessary.

5.1 Classification of structures

The classification of a structure is made according to Eurocode 3, with the critical buckling coefficient, α_{CR}. In a general way, the determination of α_{CR} comes down to a buckling analysis. Problems arise during the practical application of α_{CR}: how can the critical buckling coefficient be obtained and does it represent the appropriate buckling mode?

Comparison studies will be performed, according to Eurocode 3 and several structural programs, to determine the different buckling modes. The selected programs are ROBOT 97 and Hercule. Those programs find α_{CR} values, in an increasing order. One of those results belongs to the correct buckling mode. In the automatic procedure the lowest α_{CR} corresponding to sway buckling is used. Even though the determination of α_{CR} corresponding to non-sway buckling is on the safe side, the other elements will then not be designed as efficient as they should be.

Now the question arises whether it is possible to identify non-sway buckling. Buckling analyses yield different critical buckling coefficients with corresponding eigenvectors. Eigenvectors allow determining the deformations belonging to buckling modes. A clear distinction can be observed between non-sway and sway buckling. Non-sway buckling is when the different joints of the structure are stiff enough (nodes supposed fixed); then non-sway buckling occurs in members of the structure. Whereas sway buckling is when all nodes of the structure are "free" to displace; then sway buckling occurs when the whole structure displaces.

5.2 Modification of the model to include imperfections

To assess imperfections, Eurocode 3 proposes formulas and tables. As described in the literature survey, for the more general cases, some formulas do not seem to be applicable anymore. The automatic procedure proposes to find a way to assess frame as well as member imperfections for all cases.

In the automatic procedure first frame imperfections are applied. Eurocode 3 remarks that if the structure has already certain deformations or transverse actions it will not be necessary to assess frame imperfections (art. 5.2.4.2(2)A, NAD-Fr). Any transverse load can indeed make a non-linear analysis possible. In the automatic procedure this point will not be considered. The direction and the application of frame imperfections are based on the buckling analysis and particularly the shape of the sway buckling. The buckling analysis gives the appropriate eigenvector and buckling modes of the structure. To each node of the structure it is possible to associate proportionally defined values, resulting from the buckling analysis; then a new deformed structure is obtained.

Then, in the automatic procedure member imperfections are applied. Two possibilities can be distinguished. The first possibility (Eurocode 3): member imperfections can be avoided. For those cases not only cross-sections checks of the structure must be performed, but also stability checks of non-sway elements. Eurocode 3 allows, for pin-ended elements and elements with $\alpha_{CR,element} > 4$, member imperfections to be omitted. Though, to dimension the structure, those elements need to be checked on stability afterwards. The second possibility (automatic procedure): imperfections have to be taken into account in all elements. For those cases, according to the fact that the whole structure is deformed and stable only cross-section checks are necessary. Eurocode 3 must be used (Table 5.5.1, [EURO3,92]) to determine the value $e_{0,d}$ of member imperfections. The application of member imperfections implies the use of intermediate nodes. The buckling analysis of the structure without imperfections, with intermediate nodes, gives the necessary information on the direction to apply member imperfections. Member imperfections are applied in the direction as indicated by the buckling mode.

5.3 Non-linear analysis and elements verification

Once the structure is classified and modified (imperfections are applied to the structure) a non-linear analysis can be performed. The programs use several tools to approach the non-linear behaviour of the structure. Here the incremental method called « step by step » and the Newton – Raphson methods are employed (see 3.3.2). Those tools include imperfection influences in their calculations. After all the bending moments and forces resulting from a non-linear analysis are obtained, the different elements must be checked for strength according to Eurocode 3.

The summary of the whole procedure of the use of Eurocode 3 is represented in Figure 5.1.

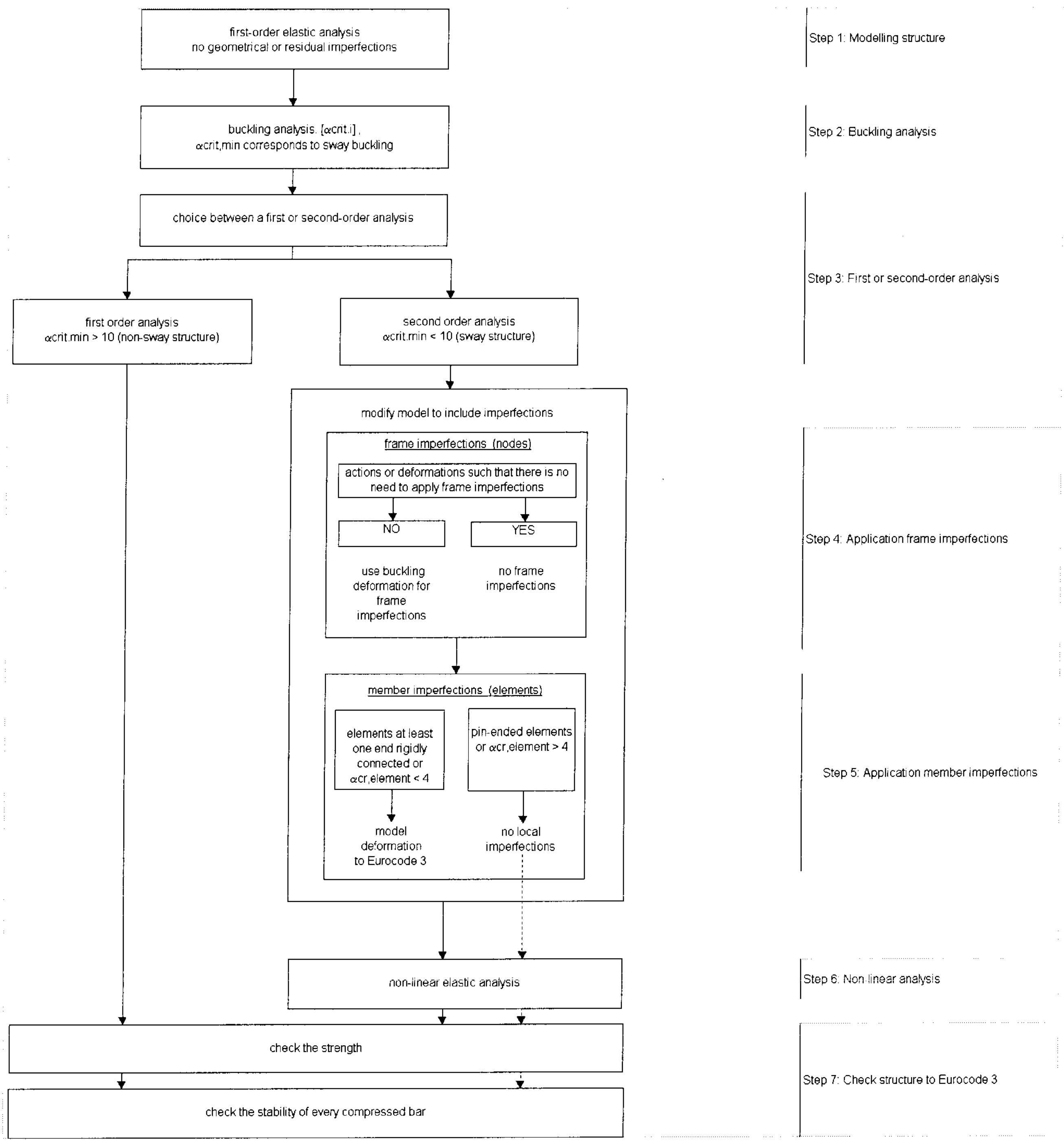


Figure 5.1: Organisation chart for an automatic procedure assessing effects of imperfections

5.4 The automatic procedure

The 7 – steps of the suggested automatic procedure are based on Eurocode 3 and are as follows:

Step 1: Modelling structure
Make a model of the structure and include intermediate nodes.

Step 2: Buckling analysis
A linear analysis determines forces and moments in each element of the structure. A buckling analysis gives a series of α_{CR} values corresponding to buckling modes. Select the lowest positive α_{CR} value for sway buckling.

Step 3: First or second-order analysis
Compare the chosen α_{CR} to the Eurocode 3 sway criteria, to decide whether or not it is necessary to apply imperfections. Two cases can be distinguished:

- $\alpha_{CR} > 10$, the structure is classified as non-sway. There is no need to apply imperfections; a first-order (linear) analysis of the structure is sufficient (already done in step 2). In that case, go directly to step 7, the strength and the stability of the structure need to be checked;
- $\alpha_{CR} < 10$, the structure is classified as sway. Imperfections need to be taken into account. Eurocode 3 distinguishes between two types of imperfections, sway and non-sway (frame and member imperfections). A second-order (non-linear) analysis of the structure needs to be carried out, go to step 4.

Step 4: Application of frame imperfections
To apply frame imperfections, the shape of the buckling deformation, resulting from a buckling analysis, corresponding to sway buckling of the structure is used. The eigenvector is proportionally modified to approach the buckling shape. Eurocode 3 does not define the size of the imperfection. Therefore, it is suggested to consider L/200 as the maximum deformation, with L being the "maximum height" of the structure. Note that this idea needs further research, but the value is suggested to be able to test the automatic procedure.

Step 5: Application of member imperfections
In the automatic procedure member imperfections need to be added to frame imperfections. The direction to apply imperfections is the one defined, using the intermediate nodes, by the sway buckling of step 2. By placing intermediate nodes, it is possible to give mathematically the directions in which each element deforms (with the eigenvector corresponding to sway buckling). The maximum deformation is defined by Eurocode 3 ($e_{0,d}$).

Step 6: Non-linear analysis
Once frame and member imperfections applied to the structure, a non-linear analysis can be performed. Second-order forces and moments are obtained.

Step 7: Check structure to Eurocode 3
In case of a first-order analysis of the structure, use the first-order forces and moments of step 2 to check the strength and the stability of the structure according to Eurocode 3. In case of a second-order analysis of the structure, use the second-order forces and moments of step 6 to check only the strength of the elements.

6. The buckling analysis of structures

The most important step to design structures, and for the automatic procedure, is the buckling analysis. It determines the critical buckling coefficient and the buckling length of structures can be deduced. Eurocode 3 considers the critical buckling coefficient α_{CR} as a main step in the design of structures. This part shows how the different programs deal with the problem of buckling analysis and how the results have to be read.

6.1 Buckling analyses applications

Among the different programs used, only two enable the determination of the coefficient α_{CR}. The programs are ROBOT97 and HERCULE. With PEPmicro it is not possible to do this type of calculation. In annexes A and B all the information concerning the use, the syntax and the way to read the results for ROBOT and Hercule are described.

6.2 The critical buckling coefficient, buckling length and discretisation in practice

In the following part, the results will be analysed. For different types of structure, the computer and manual calculations will be compared. The studied cases are:

- individual structural elements;
- single frames;
- trusses.

The relation between the critical buckling coefficient α_{CR}, buckling length L_f and discretisation is studied. To verify how the different programs react to the problem of discretisation, first the number of intermediate nodes was analysed. The following questions arise: If there are no intermediate nodes, will the buckling deformations still be correct and will the critical buckling coefficient compared with the theoretical Euler value be the same.

6.2.1 Individual structural elements

6.2.1.1 Critical buckling coefficient

The goal of the buckling analysis in the automatic procedure is to be able to know which value of the critical buckling coefficient has to be considered. The following part starts with the comparison of a pin-ended element and a fixed / pin-ended element, to study the influence of the number of intermediate nodes. Consider the two cases of Figure 6.1a and Figure 6.1b.

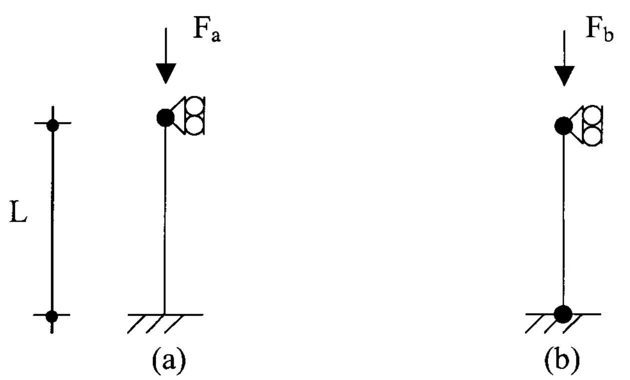


Figure 6.1: (a) fixed / pin-ended element, (b) pin-ended element

The force F_a= 250 kN and F_b= 80 kN. The elements have lengths of L = 4 m and the sections are IPE120. The axial loads are applied at the top node.

The theoretical value for the critical buckling coefficient according to Euler (α_{EULER}) is:

- for a pin-ended element:

$$\alpha_{CR} = \frac{\pi^2 \cdot EI}{L_f^2} \cdot \frac{1}{N_{SD}} = \frac{\pi^2 \cdot 2.1 \times 10^8 \cdot 318 \times 10^{-8}}{4^2} \cdot \frac{1}{80} = 5.1492$$

- for a fixed / pin-ended element:

$$\alpha_{CR} = \frac{\pi^2 \cdot EI}{L_f^2} \cdot \frac{1}{N_{SD}} = \frac{\pi^2 \cdot 2.1 \times 10^8 \cdot 318 \times 10^{-8}}{\left(\frac{4}{\sqrt{2}}\right)^2} \cdot \frac{1}{250} = 3.2955$$

The same examples were introduced in ROBOT97 and Hercule. Determining, after a buckling analysis, the critical buckling value corresponding to 0 to 5 intermediate nodes shows their influence. The results are given in Table 6.1 and graphically in Figure 6.2.

Table 6.1: α_{CR} for pin-ended element

	Pin-ended element		Pin-ended element	
	ROBOT		HERCULE	
Intermediate Nodes	α_{cr}	Δ	α_{cr}	Δ
0	6.26	21.6%	6.26	21.6%
1	5.18	0.6%	5.14	-0.1%
2	5.15	0.1%	5.11	-0.8%
3	5.15	0.0%	5.10	-0.9%
4	5.15	0.0%	5.10	-0.9%
5	5.15	0.0%	5.10	-0.9%

$$\Delta = \frac{\alpha_{FEM} - \alpha_{Euler}}{\alpha_{Euler}} \cdot 100\%$$

Where α_{EULER} = 5.15

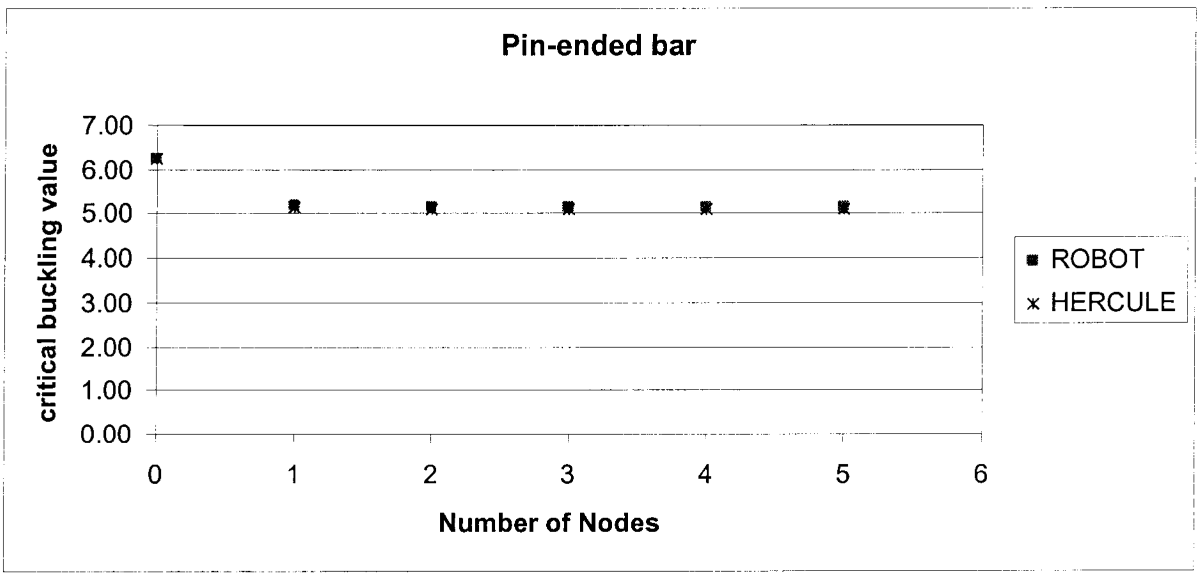


Figure 6.2: α_{CR} versus the number of nodes

From Table 6.1, it can be concluded that taking no intermediate nodes is not enough for both programs. A margin around 21.6% is obtained for both cases. For this structure only one intermediate node yields accurate results.

Note that the difference between ROBOT and Hercule is:

$$\frac{\alpha_{ROBOT} - \alpha_{HERCULE}}{\alpha_{ROBOT}} \cdot 100\% < 1\%$$

The difference is less than 1%. 1% is a very good approximation, and a few procent is considered as very good. Therefore, it can be considered that ROBOT and Hercule give similar results, for the pin-ended element.

For verification, the fixed / pin-ended element was tested under the same conditions. The results are numerically given in Table 6.2 and graphically in Figure 6.3.

Table 6.2: α_{CR} for fixed / pin-ended

	Fixed / pin-ended element ROBOT		Fixed / pin-ended element HERCULE	
Intermediate Nodes	α_{cr}	Δ	α_{cr}	Δ
0	4.97	50.6%	4.97	50.6%
1	3.43	4.1%	3.39	2.8%
2	3.37	2.3%	3.32	0.9%
3	3.36	1.9%	3.31	0.4%
4	3.35	1.8%	3.30	0.3%
5	3.35	1.7%	3.30	0.2%

$$\Delta = \frac{\alpha_{FEM} - \alpha_{Euler}}{\alpha_{Euler}} \cdot 100\%$$

Where $\alpha_{EULER} = 3.30$

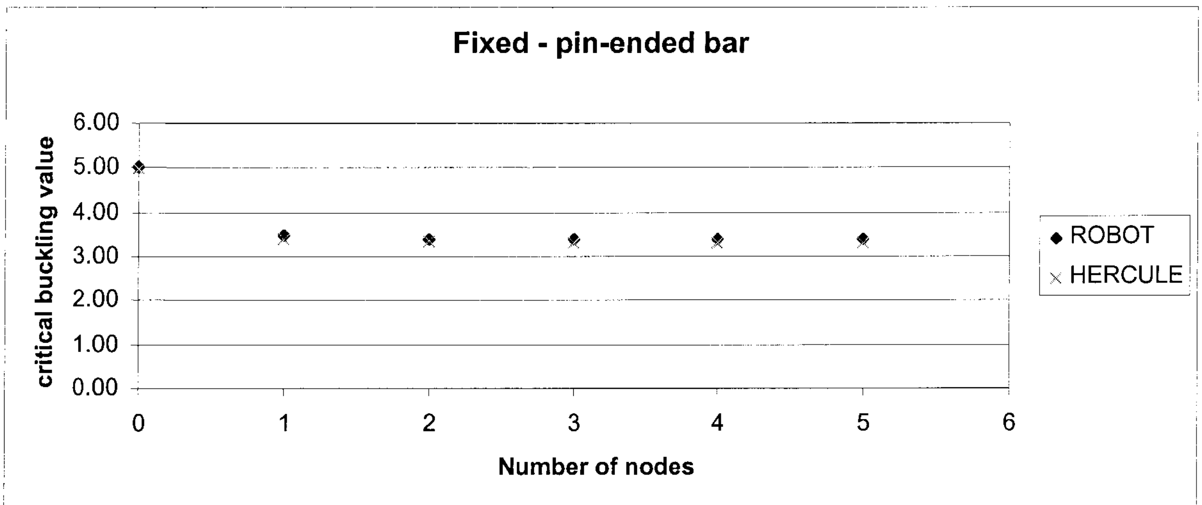


Figure 6.3: Relation between α_{CR} and the discretisation for a fixed / pin-ended element

The element does not give the same accuracy as the pin-ended element. The convergence is not as fast as the pin-ended element. Using one intermediate node, a good approximation of the result is obtained: 4.1 % for ROBOT97 and 2.8 % for Hercule. It seems like, for this case, Hercule gives a better approximation of the results.

The difference between the programs is:

$$\frac{\alpha_{ROBOT} - \alpha_{HERCULE}}{\alpha_{ROBOT}} \cdot 100\% = 1.7\%$$

As for the pin-ended element, a convergence can be noted after one intermediate node for ROBOT and Hercule. The automatic procedure suggests considering the application of member imperfections even to pin-ended element. The results show that at least three intermediate nodes is necessary to obtain a very good accuracy for ROBOT and for Hercule. Different tests and the literature showed that four intermediate nodes gives a better accuracy of the results, in case of a non-linear analysis of more complex structures.

6.2.1.2 Maximum number of modes

For the previous examples, the maximum number of modes possible were checked. The results are represented in Table 6.3.

Table 6.3: Number of buckling modes depending on the number of intermediate nodes

	ROBOT et Hercule	
	Number of modes possible	
Intermediate nodes	Pin-ended element	Fixed / pin-ended element
0	2	1
1	4	3
2	6	5
3	8	7
4	10	9
5	12	11

From Table 6.3, we can conclude that adding a node seems to allow two supplementary buckling modes. Within one finite element, only double curvature can be described resulting in the buckling modes of Table 6.3, which are represented by Figure 6.4 for zero and one intermediate nodes.

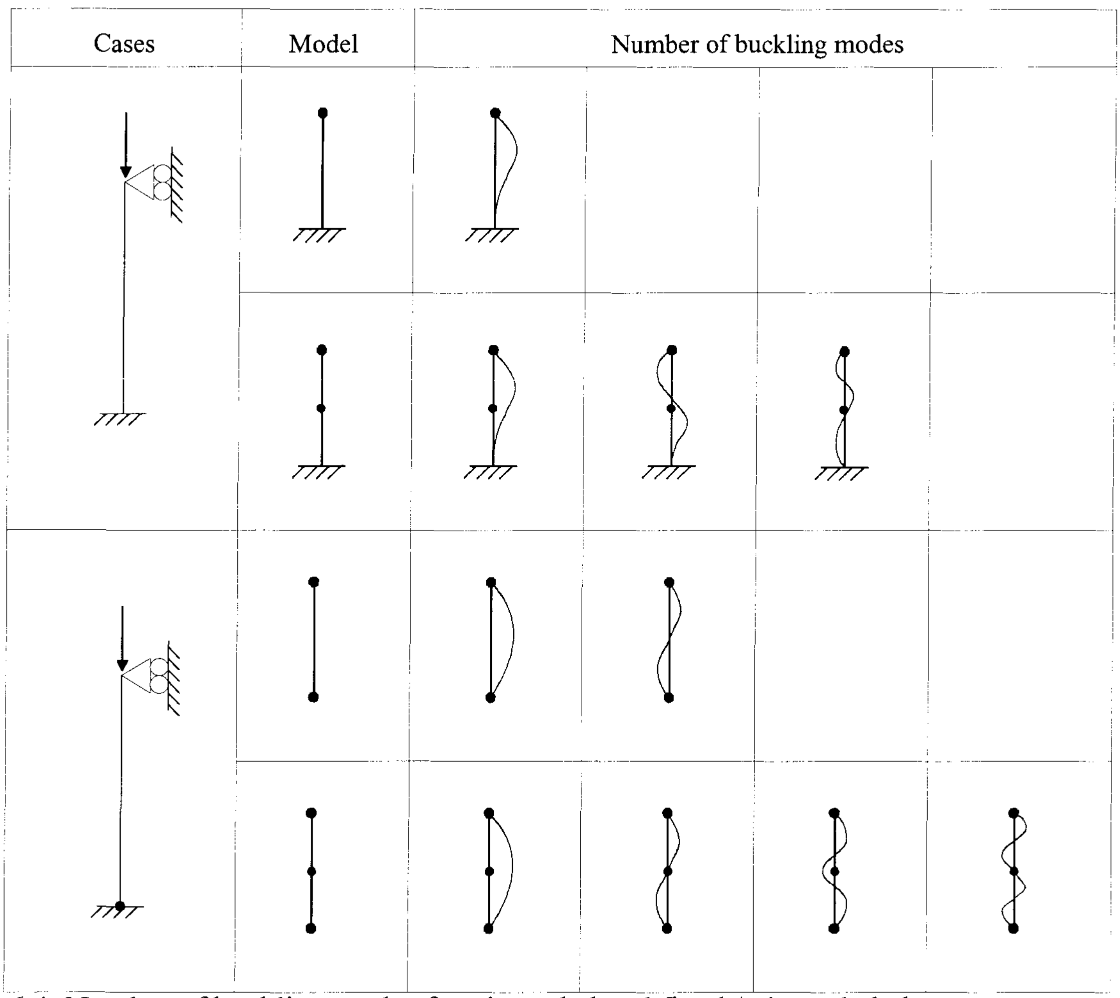


Figure 6.4: Number of buckling modes for pin-ended and fixed / pin-ended elements

6.2.1.3 Buckling deformations

Apparently, the previous applications show that even though there are no intermediate nodes, the form of the buckling deformation is still correctly represented.

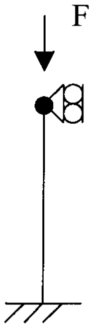


Figure 6.5: Fixed / pin-ended element

Consider the force F = 250 kN represented in Figure 6.5. The element has a length of 4m and the section is IPE120. The axial load is applied at the top node. The top node is free to move along the vertical axis.

The buckling deformations from ROBOT and Hercule are represented in Figure 6.6. The results obtained are identical.

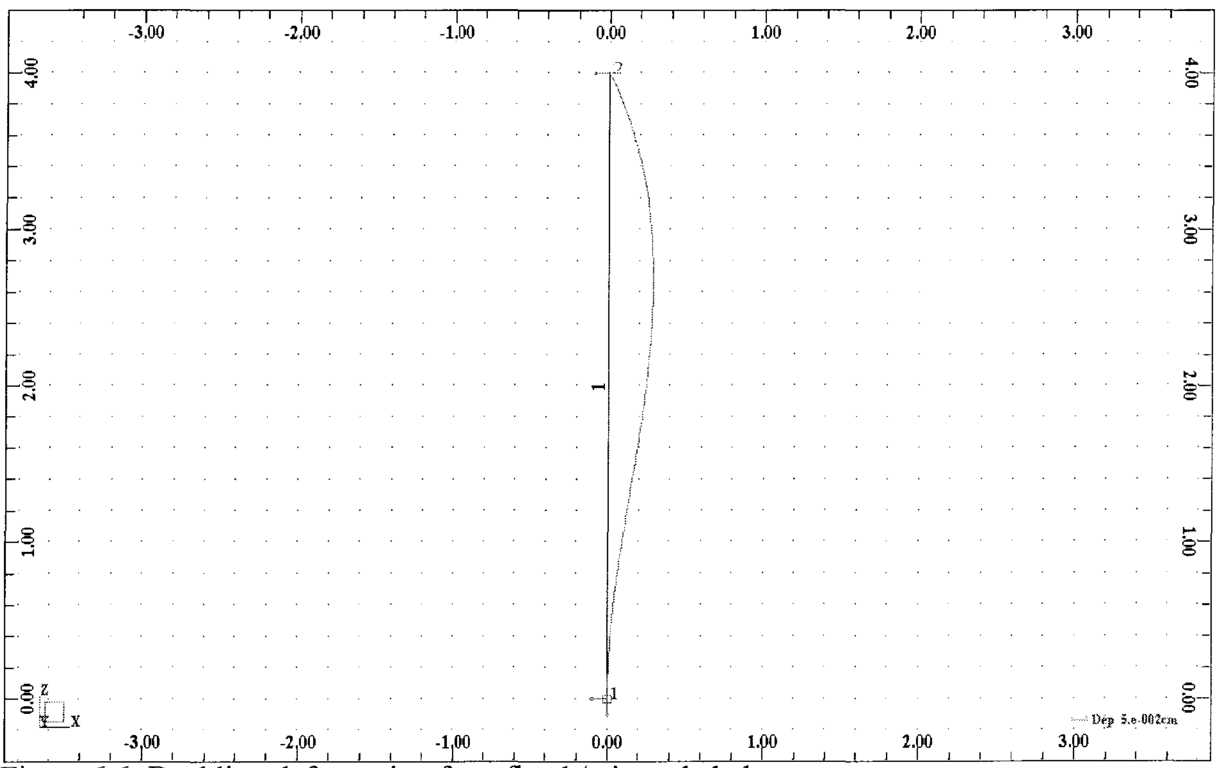


Figure 6.6: Buckling deformation for a fixed / pin ended element

During the calculation process, the program assembles stiffness matrices $[K_0]$ and $[K_G]$ and determines the solution of: det $([K_0] + \lambda\ [K_G]) = 0$. In our case, without intermediate nodes the stiffness matrix has very few terms. The two nodes and the degrees of freedom of the structure only allow one buckling mode.

The computer model of that element has two degree of freedom UZ and RY at the top node. The eigenvectors are represented in Table 6.4. It shows only a rotation at the top node.

Table 6.4: Eigenvector for a fixed / pin-ended element

Node	Case	Eigenvector UX	Eigenvector UZ	Eigenvector RY
1	1	0	0	0
2	1	0	0	-0.0027

The programs ROBOT and Hercule smooth the form of the buckling deformation based on node rotations, RY. Those buckling modes are actually approximations of the reality; no exact values of the eventual position of intermediate nodes on the curve are given.

Placing additional nodes gives extra information. After a buckling analysis, the eigenvector will have terms at every node of the structure. The proportional relation between the displacements of the nodes will correspond to a certain buckling mode. The magnitude of buckling deformation is unknown.

Remark 1: Tests have been carried out to check the influence of the form of the element on the shape of the buckling deformation. Each element has three intermediate nodes. The form of the imperfection, represented in Figure 6.7, is according to:

- a straight element;
- a sinusoidal element;
- a double curved element.

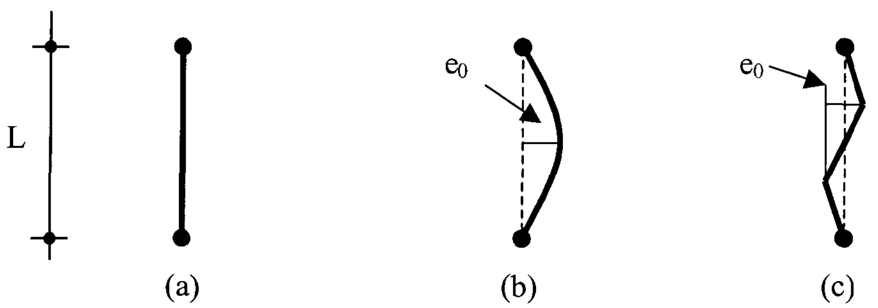


Figure 6.7: Form of imperfection: straight (a), simple arc (b) double arc (c).

For the three different cases, the same first mode of buckling deformation was found. For pin-ended elements, the form of the imperfection does not affect the shape of the buckling deformation.

Remark 2: Comparative tests were carried out on inclined and vertical elements. It was shown that the buckling deformations in both cases are the same.

In the following part, a single frame will be studied. The influence of intermediate nodes will be tested again. The buckling length will be determined following codes and methods.

6.2.2 Single frames

The relationship between the buckling analysis, buckling length and discretisation is now investigated for two single frames. First a single frame with fixed joints will be used to determine the buckling length according to several methods and codes. Then a single frame with one pin-ended vertical element is used for a computer buckling analysis.

6.2.2.1 Single frames and buckling length

In this part the precision of the different buckling lengths determined with several codes and programs is verified. The methods and programs are:

- Eurocode 3, annexe E;
- the French code, CM66;
- ROBOT;
- HERCULE.

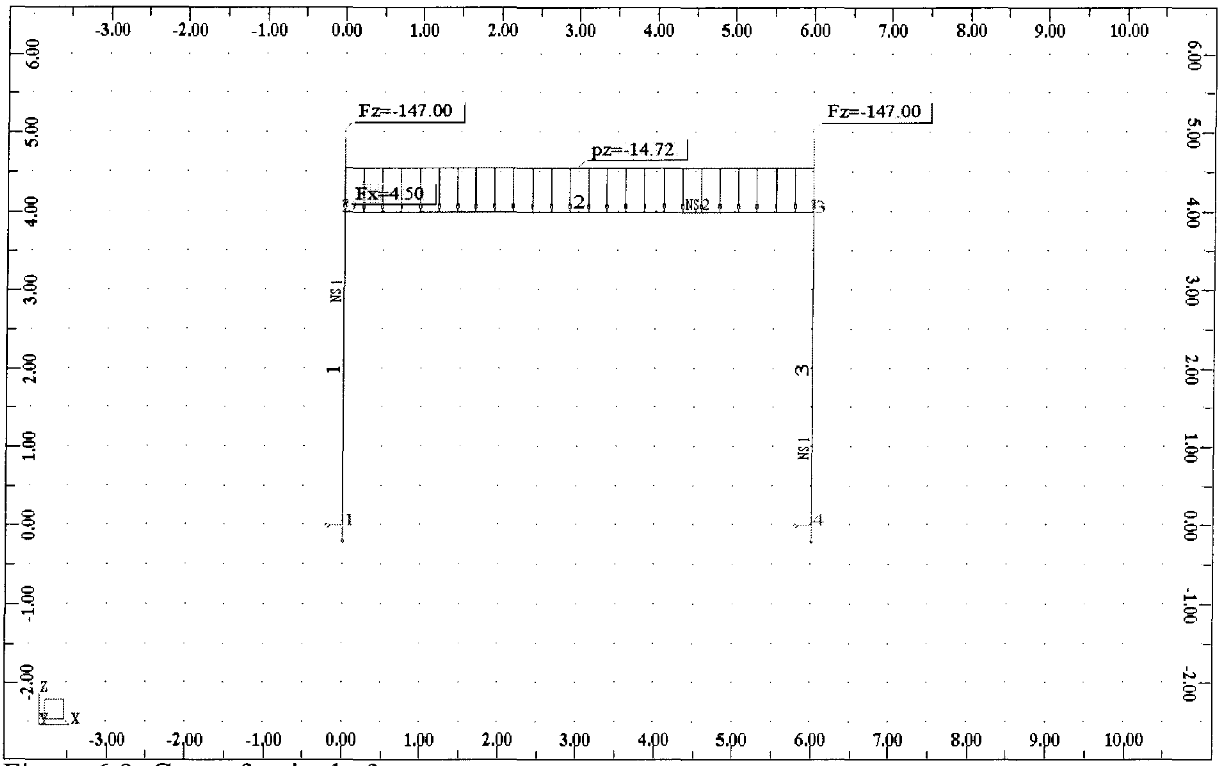
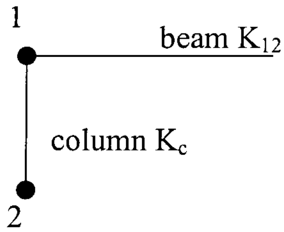


Figure 6.8: Case of a single frame

The frame is represented in Figure 6.8. The vertical elements 1 and 3 are NS1 = HEA220 and the horizontal element 2 is NS2 = IPE 200. The support nodes 1 and 4 are pin-ended. There are two vertical loads at nodes 2 and 3, F_Z = 147 kN, one horizontal load at node 2, F_X = 4.5 kN, and a distributed load on element 2, P_Z = 14.72 kN/m.

- Determination of the buckling length according to the (K_A, K_B) method based on annexe E of Eurocode 3:



The buckling length for this example is determined for a sway mode.

1) Stiffness of column HEA 220:

$$K_C = \frac{I}{L} = \frac{0.5410 \cdot 10^{-4}}{4} = 0.1353 \cdot 10^{-4} m^3$$

2) Determination of the effective stiffness at node 1:
The beam has the following stiffness:

$$K_{12} = \frac{I}{L} = \frac{0.1943 \cdot 10^{-4}}{4} = 0.4847 \cdot 10^{-5} m^3$$

It is then possible to define:

$$\eta_1 = \frac{K_C}{K_C + K_{12}} = \frac{0.5410 \times 10^{-4}}{0.5410 \times 10^{-4} + 0.4847 \times 10^{-5}} = 0.7362$$

3) Determination of the effective stiffness at node 2:
Node 2 is pin-ended that means that:

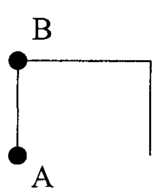
$$\eta_2 = 1$$

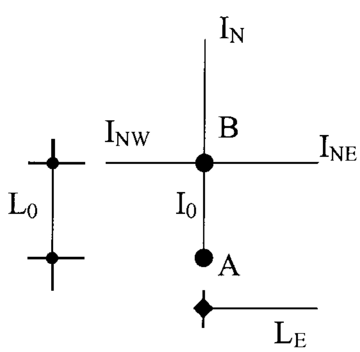
4) Determination of the buckling length:

$$l = L\left[\frac{1 - 0.2(\eta_1 + \eta_2) - 0.2\eta_1\eta_2}{1 - 0.8(\eta_1 + \eta_2) + 0.6\eta_1\eta_2}\right]^{0.5} = 4\left[\frac{1 - 0.2(0.7362 + 1) - 0.12 \times 1 \times 0.7362}{1 - 0.8(0.7362 + 1) - 0.6 \times 1 \times 0.7362}\right]^{0.5} = 13.08m$$

- Determination of the buckling length based on the CM66:

1) The general case:
This method looks like the one from Eurocode 3, but is slightly different. For the same example we have:





For the frame, the stiffness of point B is:

$$K_B = \frac{\frac{I_{NE}}{L_E}}{\frac{I_{NE}}{L_E} + \frac{I_0}{L_0}} = \frac{\frac{0.1943 \cdot 10^{-4}}{6}}{\frac{0.1943 \cdot 10^{-4}}{6} + \frac{0.5410 \cdot 10^{-4}}{4}} = 0.193$$

The point A is pin-ended, so $K_A = 0$

The structure is sway, so the buckling length becomes (art.5,134):

$$\frac{L_f}{L} = \sqrt{\frac{1.6 + 2.4(K_A + K_B) + 1.1K_A K_B}{K_A + K_B + 5.5K_A K_B}}$$

$$\frac{L_f}{L} = \sqrt{\frac{1.6 + 2.4(K_B)}{K_B}} = \sqrt{\frac{1.6 + 2.4(0.193)}{0.193}} = 3.268$$

$$L_f = 3.268 \cdot 4 = 13.07m$$

2) Simplified case:
The second moment of area I_m and the length l_m are concerning the vertical elements and I_t and l_t, concerning the horizontal element.

CM 66 (art. 15,134-2) defines:

$$\frac{I_m \cdot l_t}{l_m \cdot I_t} = \frac{0.541 \cdot 10^{-4} \times 6}{4 \times 0.1943 \cdot 10^{-4}} = 4.1765 > 4$$

so that

$$\frac{l}{L} = 2\sqrt{1 + 0.4 \cdot \frac{I_m \cdot l_t}{l_m \cdot I_t}} = 2\sqrt{1 + 0.4 \cdot 4.1765} = 3.268$$

$$l = 13.07m$$

Conclusion
For both methods of CM66, general and simplified, the same value of buckling length is obtained. The difference between Eurocode 3 and CM66 is:

$$\frac{L_{f,EC3} - L_{f,CM66}}{L_{f,CM66}} \cdot 100\% = \frac{13.08 - 13.07}{13.07} \cdot 100 = 0.07\%$$

The results of Eurocode 3 and CM66 differ by 0.07%. The results are considered the same. To verify the results of the different programs, the selected value of the buckling length is:
$L_{f,CODE} = 13.07$ m

Remark: Note that for the two previous methods, the determination of the buckling length only takes into account the geometry, not the applied loads.

- Determination of the buckling length with the program ROBOT97:
  The buckling length, for ROBOT97, is obtained after a buckling analysis of the structure. Within the program the following formula is used:

$$\alpha_{CR} = \frac{\Pi^2 \cdot EI}{L_f^2} \cdot \frac{1}{N_{SD}} \Leftrightarrow L_f = \sqrt{\frac{\Pi^2 \cdot EI}{\alpha_{CR}} \cdot \frac{1}{N_{SD}}}$$

  For the previous example, the buckling analysis with ROBOT97 gives directly the following results on request:
  $\alpha_{CR} = 3.51$
  $L_f = 13.01$ m

- Determination of the buckling length with the program Hercule:
  The program Hercule gives directly the value of the coefficient α_{cr} = 3.43, but not the buckling length L_f. It is necessary to do an external calculation to determine the buckling length:

$$L_f = \sqrt{\frac{\Pi^2 \cdot EI}{\alpha_{CR}} \cdot \frac{1}{N_{SD}}} = \sqrt{\frac{\Pi^2 \cdot 2.1 \times 10^8 \cdot 5410 \times 10^{-8}}{3.43} \cdot \frac{1}{194.16}} = 12.97m$$

Recapitulation of the results

The selected code value is $L_{f,\text{CODE}}$ = 13.07 m, both codes gave the same results. The results from ROBOT97 and Hercule are not that close, but very good. Table 6.7 shows that the difference Δ between the code and the programs is less than 1%. And the difference between the programs is:

$$\frac{L_{f,HERCULE} - L_{f,ROBOT}}{L_{f,ROBOT}} \cdot 100\% = 3‰$$

Therefore, the results from manual and computer calculations are considered to be equal. Not placing any intermediate nodes, for this example, had little influence on the results. Table 6.7 represents the different solutions corresponding to 0 and 1 intermediate node.

Table 6.7: Buckling lengths

	ROBOT 97		Hercule	
Intermediate Nodes	L_f (m)	Δ	L_f (m)	Δ
0	13.01	-0.5%	12.97	-0.8%
1	13.02	-0.4%	12.98	-0.7%
$\Delta = \frac{L_{f,FEM} - L_{f,CODE}}{L_{f,CODE}} \cdot 100\%$		Where $L_{f,\text{CODE}}$ = 13.07 m		

6.2.2.2 Single frames and eigenvectors

For all structures, the buckling analysis enables the determination of the critical buckling coefficient with their corresponding buckling modes. The programs ROBOT and Hercule do determine these α_{CR} but do not order them the same way. ROBOT orders the values in an increasing order, whereas Hercule classifies the absolute values in an increasing way, adding afterwards the sign. From the results for α_{CR} only the one corresponding to elements under compression are considered; they are the positive ones: For the automatic procedure the important critical buckling coefficient α_{CR} is the one that corresponds to sway buckling of the structure.

The considered structure is now a single frame with one pin-ended vertical element. After a description of the structure, three particular cases will be considered. First, Case 1 has an IPE100 as pin-ended vertical element. Then, case 2 same loading but the pin-ended vertical element becomes a HEA200. Finally, Case 3 the same structure is considered but with greater vertical force on the pin-ended vertical element.

Figure 6.9 represents the structure. The support nodes 1 and 7 are pin-ended; there is a hinge at element 5 at the side of node 5. The three cases are:

- the elements 1 to 4 are HEA 220, and the elements 5 and 6 are IPE 100. The loading is the one represented in Figure 6.9;
- the elements 1 to 4 still are HEA 220 and the elements 5 and 6 are HEA 200. The loading is still the one represented in Figure 6.9;
- the elements 1 to 4 still are HEA 220, and the elements 5 and 6 still are HEA 200. The loading on node 3 remains F_{Z1} = -340 kN, but the one for node 5 is increased, F_{Z5} = -270 kN.

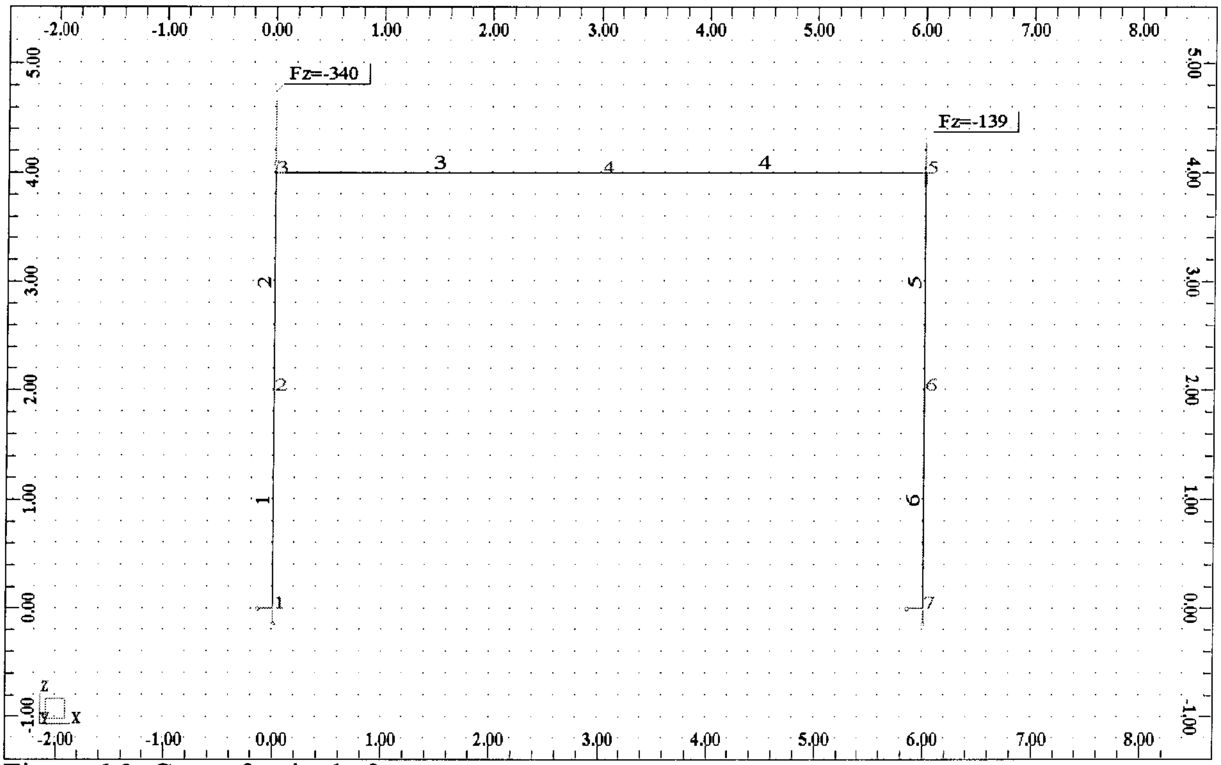
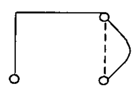
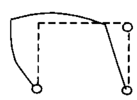
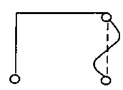


Figure 6.9: Case of a single frame

The results:
A buckling analysis of the structure is made. Table 6.8 represents the eigenvectors corresponding to the three lowest critical buckling coefficients and the corresponding buckling modes for case 1.

Case 1: Single frame with one vertical pin-ended element IPE 100

Table 6.8: Buckling results frame 1

Case 1		Eigenvectors								
		Mode 1			Mode 2			Mode 3		
	Nodes	UX	UZ	Ry	UX	UZ	Ry	UX	UZ	Ry
	1	0	0	0	0	0,0009	0	0	0	0
	2	0	0	0	0,0017	0	0,0007	0	0	0
	3	0	0	0	0,0029	0	0,0004	0	0	0
	4	0	0	0	0,0029	-0,0005	-0,0001	0	0	0
	5	0	0	0	0,0029	0	-0,0002	0	0	0
	6	0,0025	0	-0,0001	0,0014	0	0,0007	-0,0004	0	-0,0026
	7	0	0	0,002	0	0	0,0007	0	0	0,0029
α_{cr}	Hercule	1,600			1,714			8,493		
	ROBOT	1,659			1,733			9,896		
Buckling modes										

Regarding Table 6.8, the following remarks can be made:
- non-sway buckling occurs before sway buckling. Indeed for the first mode, it is the pin-ended element that buckles before the rest of the structure;
- non-sway buckling, the eigenvectors of mode 1 show that values appear only for elements that buckle. For instance:
  - the values are zero for the elements 1 to 4 (nodes 1 to 5) indicating that nothing happened in those elements;
  - the other values indicate either a displacement (U_X,U_Z) or a rotation of the node (R_Y).
- note that for node 5, for modes 1 and 3 of Table 6.8, the eigenvectors are equal to zero. Whereas, the buckling modes show a clear rotation at node 5. Apparently the given eigenvectors correspond to the horizontal element and not to the vertical element of the structure. That means, for the automatic procedure, that the whole element information has to be check to avoid this problem;
- another surprising value, is the eigenvector UX at node 6 of Table 6.8. Looking at the buckling mode, the value zero is expected. This value is clearly smaller than the one from mode 1 and 2, but not zero. It looks like during the solving of the determinant a small numeric imperfection appeared at that node;
- the values obtained for the nodes enable the program to smooth the shape of the buckling deformation. But the exact values of those displacements are not given. The values of Table 6.8 are only relative values. The absolute values corresponding to the buckling modes are unknown. The different programs normalised these values to draw the buckling deformation;
- the values of Table 6.9 are the one from Hercule. Note that even though the values are not the same for the nodes, the same buckling deformation is drawn.

Table 6.9: Results of the eigenvector with Hercule.

```
**********************************************************************************
*                                                    *
*                     CAS   101                      *
*                                                    *
* MODE PROPRE DE  FLAMBEMENT  NUMERO   2             *
*                                                    *
**********************************************************************************
```

Structure active : *Portique articulé*

*	NOEUD	*	X	Y	Z	*	RX	RY	RZ	*
* DEPLACEMENT	1(0)	*	0.000000000	0.000000000	0.000000000	*	0.000000000	0.000000000	0.000000000	*
* DEPLACEMENT	2(0)	*	0.000000000	0.000000000	0.000000000	*	0.000000000	0.000000000	0.000000000	*
* DEPLACEMENT	3(0)	*	0.000000000	0.000000000	0.000000000	*	0.000000000	0.000000000	0.000000000	*
* DEPLACEMENT	4(0)	*	0.000000000	0.000000000	0.000000000	*	0.000000000	0.000000000	0.000000000	*
* DEPLACEMENT	5(0)	*	0.000000000	0.000000000	0.000000000	*	0.000000000	0.000000000	0.000000000	*
* DEPLACEMENT	6(0)	*	-0.061087465	0.000000000	0.000000000	*	0.000000000	0.000000102457	0.000000000	*
* DEPLACEMENT	7(0)	*	0.000000000	0.000000000	0.000000000	*	0.000000000	-0.047585745	0.000000000	*

23 Juin 1999 13:17:04

****
* *
* CAS 102 *
* *
* MODE PROPRE DE FLAMBEMENT NUMERO 2 *
* *
****

Structure active : *Portique articulé*

* NOEUD		*	X	Y	Z	*	RX	RY	RZ	*
* DEPLACEMENT	1(0)	*	0.000000000	0.000000000	0.000000000	*	0.000000000	-0.020785616	0.000000000	*
* DEPLACEMENT	2(0)	*	-0.039945639	0.000000000	-0.000013972	*	0.000000000	-0.017965614	0.000000000	*
* DEPLACEMENT	3(0)	*	-0.068940666	0.000000000	-0.000027944	*	0.000000000	-0.010077031	0.000000000	*
* DEPLACEMENT	4(0)	*	-0.068949790	0.000000000	0.011282582	*	0.000000000	0.001132299	0.000000000	*
* DEPLACEMENT	5(0)	*	-0.068958915	0.000000000	0.000174448	*	0.000000000	0.004868742	0.000000000	*
* DEPLACEMENT	6(0)	*	-0.034479457	0.000000000	0.000087224	*	0.000000000	-0.017239729	0.000000000	*
* DEPLACEMENT	7(0)	*	0.000000000	0.000000000	0.000000000	*	0.000000000	-0.017239729	0.000000000	*

23 Juin 1999 13:17:04

****
*   *
*   CAS 103   *
*   *
* MODE PROPRE DE FLAMBEMENT NUMERO 3   *
*   *
****

Structure active : *Portique articulé*

* NOEUD	*	X	Y	Z	*	RX	RY	RZ	*
* DEPLACEMENT	1(0) *	0.000000000	0.000000000	0.000000000	*	0.000000000	0.000000000	0.000000000	*
* DEPLACEMENT	2(0) *	0.000000000	0.000000000	0.000000000	*	0.000000000	0.000000000	0.000000000	*
* DEPLACEMENT	3(0) *	0.000000000	0.000000000	0.000000000	*	0.000000000	0.000000000	0.000000000	*
* DEPLACEMENT	4(0) *	0.000000000	0.000000000	0.000000000	*	0.000000000	0.000000000	0.000000000	*
* DEPLACEMENT	5(0) *	0.000000000	0.000000000	0.000000000	*	0.000000000	0.000000000	0.000000000	*
* DEPLACEMENT	6(0) *	0.001633815	0.000000000	0.000000000	*	0.000000000	0.027647948	0.000000000	*
* DEPLACEMENT	7(0) *	0.000000000	0.000000000	0.000000000	*	0.000000000	-0.028403439	0.000000000	*

The last remark offers two possibilities:

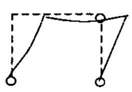
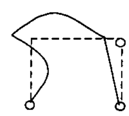
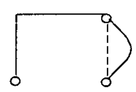
1) Both the programs do not work in the same number of dimension.
   ROBOT works in a two-dimensional plane whereas Hercule works in three-dimensional space. In Hercule it is necessary to eliminate certain components; that can modify the accuracy of the results.

2) There is a difference in the tools used to solve the determinant.
   Both the programs use the following characteristic determinant:
   $\det (K_0 + \lambda . K_\sigma) = 0$
   A difference can exist in the tools used to solve the determinant; ROBOT and Hercule use a subspace iteration method. The differences in the subspace can influence the accuracy of the results.

Case 2: Single frame with one vertical pin-ended element HEA 200

As case 1, Table 6.10 represents the eigenvectors corresponding to the three lowest critical buckling coefficients and the corresponding buckling modes for case 2. The frame has the same loading as case 1, but the vertical pin-ended element is modified.

Table 6.10: Buckling results frame 2

Case 2		Eigenvectors								
		Mode 1			Mode 2			Mode 3		
	Nodes	UX	UZ	Ry	UX	UZ	Ry	UX	UZ	Ry
	1	0	0	0,0009	0	0	0,0012	0	0	0
	2	0,0017	0	0,0007	0,0013	0	-0,0002	0	0	0
	3	0,0029	0	0,0004	-0,0004	0	-0,0010	0	0	0
	4	0,0029	-0,0005	-0,0001	-0,0004	0,0011	0,0001	0	0	0
	5	0,0029	0	0,0002	-0,0004	0	0,0005	0	0	0
	6	0,0014	0	0,0007	-0,0002	0	-0,0001	0,0025	0	-0,0001
	7	0	0	0,0007	0	0	-0,0001	0	0	0,002
α_{cr}	Hercule	1,716			25,712			33,29		
	ROBOT	1,736			27,726			35,827		
Buckling modes										

As expected, for the same loading, the stiffer (EI) the structure has, the higher the critical buckling coefficient (α_{cr}) is. From the literature survey, it is known that the relation between the critical load V_{CR} and the total vertical load V is:

$$\alpha_{CR} = \frac{V_{CR}}{V} \text{ where } V_{CR} = \frac{\pi^2 \cdot EI}{L_F^{\,2}}$$

The bucking mode 1 of case 2, represented in Table 6.10, comes very close to the buckling mode 2 represented in Table 6.8 of case 1. In both cases it represents sway buckling. The remarks from case 1 are still valid for case 2. But due to the increasing second moment of area of the pin-ended element, the first buckling mode is a sway buckling mode.

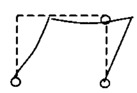
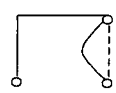
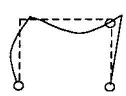
It can be concluded that it is possible, using the eigenvectors, to identify sway buckling. Therefore, eigenvectors have to comply with the following points:

- the minimum positive buckling value must be considered;
- each displacement of the eigenvector at the nodes, other than the support nodes, must show a non-zero value.

Case 3: A single frame equivalent to case 2, but with greater force on the pin-ended element.

Case 3 is used to point out one of Eurocode 3 clauses on buckling length. Table 6.11 represents the eigenvectors corresponding to the three lowest critical buckling coefficients and the corresponding buckling modes for case 3.

Table 6.11: Buckling results frame 3

Case 3		Eigenvectors								
		Mode 1			Mode 2			Mode 3		
	Nodes	UX	UZ	Ry	UX	UZ	Ry	UX	UZ	Ry
	1	0	0	0,0008	0	0	0	0	0	0,0012
	2	0,0015	0	0,0007	0	0	0	0,0014	0	-0,0002
	3	0,0025	0	0,0004	0	0	0	-0,0003	0	-0,0010
	4	0,0025	0,0004	0	0	0	0	-0,0003	0,0012	0,0001
	5	0,0025	0	-0,0002	0	0	0	-0,0003	0	0,0005
	6	0,0013	0	0,0006	0,0018	0	0	-0,0002	0	-0,0001
	7	0	0	0,0006	0	0	0,0014	0	0	0,0001
α_{cr}	Hercule	1,354			17,138			25,598		
	ROBOT	1,370			18,440			27,600		
Buckling modes										

As expected, by increasing the value of the total vertical load, the structure's critical buckling coefficient decreases from 1.71, resulting from mode 2 of case 1 and mode 1 of case 2, to 1.35, resulting from mode 1 of case 3. The only way to obtain the same value of the critical buckling coefficient is to have a constant total vertical load. The distribution of the total load over the structure does not have an influence on the critical buckling coefficient, but the buckling lengths of the different compressed elements do change.

The two frames have forces applied on the vertical elements. Between the case 2 and 3, the force above the pin-ended element is amplified by a factor 1.94. According to Eurocode 3, when defining the buckling length, only the geometry of the structure is taken into account and not the loading. For the cases 2 and 3 the geometry is identical, so theoretically the same value of buckling length should be obtained.

It can be concluded that, first of all the distribution of the total vertical load does not influence the critical buckling coefficient. The buckling length depends only on the axial load on the different elements, and not on any transverse loads. Changing the load distribution on the structure implies a variation of the buckling length in each element. Eurocode 3 does not include load distribution effects when calculating buckling length. It can be concluded as well that the buckling analysis gives a series of values for α_{CR}. It is possible, with the eigenvectors, to find the α_{CR} corresponding to sway buckling of the structure.

The problems surrounding the critical buckling coefficient were pointed out in this part. Next, different structures will be studied to verify the previous conclusions.

6.2.3 Trussed portal frames

Two different structures are considered, in this part, to verify the previous conclusions from the individual structural elements and the single frames. In particular, the influence of placing an intermediate node and the way to interprete the results from a buckling analysis.

Consider the two structures represented in Figure 6.10a and Figure 6.10b. The structures can be subdivided into two systems. The first system is the truss itself and the second one is a structure represented by a single element with infinite second moment of area, subjected to a vertical load. A pin-ended element links those two systems, so the truss supplies the sway rigidity to the structure. The infinite second moment of area implies that it will be the truss, and not the structure next to the truss, that will buckle first.

The loading on both structures is applied the same way, only the applied forces of structure 2 are lower than the ones of structure 1. The support points 1,15 and 26 are pin-ended.
Structure 1 has the following loading:

- a horizontal force at node 3:
  Fx = 70 kN
- a series of vertical forces on nodes 3 to 13:
  Fy = 50 kN
- a vertical force at node 25:
  Fy = 2500 kN

Element 45 has an infinite second moment of area. The "columns" are HEA500. The horizontal chords of the truss are HEA240. The braces are round tubes. Element 44 is HEA340.

The structure 2 has the following loading:

- a horizontal force at node 3:
  Fx = 55 kN
- a series of vertical actions on nodes 3 to 13:
  Fy = 30 kN
- a vertical force at node 25:
  Fy = 2500 kN
- a vertical force at node 13:
  Fy = 500 kN

Element 45 has an infinite second moment of area. The "columns" are SHS 400*10. The horizontal chords of the truss are HEM120. The braces are T-pieces with different dimensions. Element 44 is HEA340. The element 46 is a tube.

To pre-dimension the structures, a linear analysis without intermediate nodes has been performed with ROBOTv6. The forces are used in SEERROB, an internal program of INGEROP – CLFD (a post-processor of ROBOTv6). Then the elements and sections are verified by BAR, an internal program of INGEROP – CLFD (verification program for steel elements).

After pre-dimension ing, the structures are introduced in ROBOT and Hercule. The main characteristics were introduced manually. And the reactions at the supports were verified.

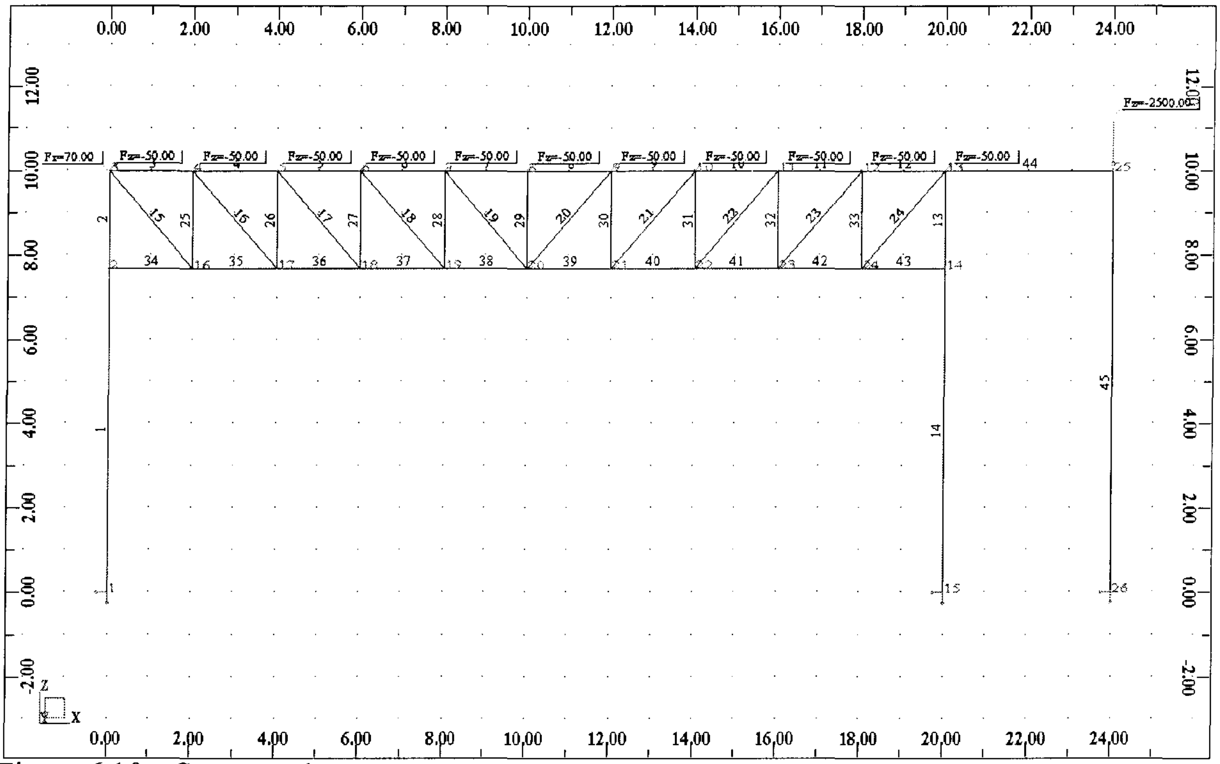


Figure 6.10a: Structure 1

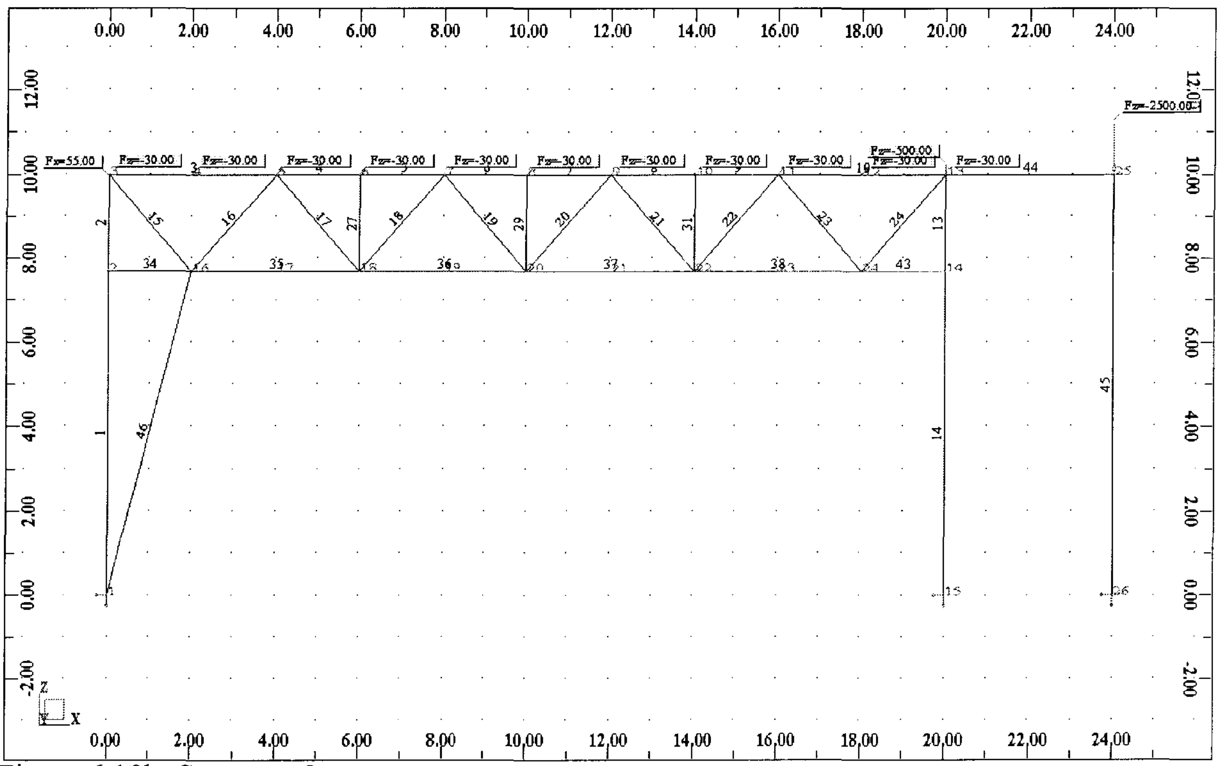


Figure 6.10b: Structure 2.

In the force diagram of structure 1, especially in the axial forces diagram, only the upper chord of the truss and the vertical elements were compressed. A second structure was then designed. The slenderness of the different elements is around 120. Due to this slenderness, an element had to be added to the structure (the element 46) so that the structure could pass the Serviceability Limit State. The diagonals were placed in «V» shape. That resulted in one compressed diagonal out of two. Non-sway buckling determines then the behaviour when compared to sway buckling. This is in order to recognise this problem to more complex cases. The forces had to be significant; therefore only three vertical elements were placed in the truss.

The case without intermediate node has been studied graphically. Each mode was observed; the results and comments are indicated in Table 6.12 and Table 6.13. The results show that some values of the critical buckling coefficient correspond to non-sway buckling. There was only one mode that corresponds to sway buckling, mode 7 for ROBOT and mode 1 for Hercule.

Note that both the programs ROBOT and Hercule show also negatives values for α_{CR}. Those values correspond to elements under tension. For an automatic procedure, these values must be ignored. The critical buckling coefficient to classify the structure is the one that belongs to sway deformation. As described earlier for a single frame, it is not possible to consider only the smallest value, because the possibility remains that non-sway buckling appears first (see case 1 of single frames).

The way to isolate non-sway buckling as described for the single frame is still valid. Graphically, sway deformation happens when, for a particular value of the coefficient $\alpha_{CR,}$ all the nodes of the structure have a displacement in the plane of the structure. Consider the eigenvector corresponding to the different critical buckling coefficients, it will be observed that the values of the displacements of the different nodes will be small, except for the eigenvectors corresponding to sway deformation, where the displacements of the nodes are not negligible anymore.

In an automatic procedure, to consider the value of the critical buckling coefficient α_{CR}:
- eliminate all the negative values of the results. Because they correspond to the elements under tension;
- find the eigenvectors corresponding to the mode where the displacements in the plane can not be neglected anymore.

Until now, leaving out intermediate nodes still yields sway or non-sway buckling modes. Different tests showed that the programs use the rotation at the end of the elements to smooth the buckling deformation. Also it was found that placing an intermediate node improved the precision of the results. In an automatic procedure four intermediate nodes allow a good precision of the critical buckling coefficient, α_{CR}. But it also means that more buckling modes have to be considered to obtain the sway buckling. Even for structures like the two trusses, as a result of placing extra intermediate nodes, it also means more buckling modes in the output file. This is valid for ROBOT and Hercule. There are practical cases where placing an extra node is necessary, e.g. the modelling of heavy point loads. The pin-ended elements are elements where, for instance, it is not necessary to apply any extra nodes. Those elements are studied in a later phase in the procedure.

Now the different problems surrounding the buckling analysis, i.e. the critical buckling coefficient, the buckling length and the discretisation, are pointed out. Several methods were suggested to solve those problems. The automatic procedure can be tested on two examples: an individual structural element and a single frame.

Table 6.12: Results Hercule without intermediate nodes

Hercule, case of a structure without intermediary nodes			
Mode	α_{cr}	Remarks The buckling deformation:	Eigenvalue:
1	4,67	GLOBAL	rotations of the columns, horizontal and vertical displacements of the truss nodes
2	- 11,00	chords	rotations at the lower chord, small horizontal displacements
3	12,15	upper chord node 4	no horizontal displacement, displacement of node 4 of the upper chord
4	- 19,31	chords	rotations at the lower chord, small horizontal and vertical displacements
5	30,76	upper chord	rotations at the upper chord, small horizontal displacements
6	- 33,00	truss	double curve of the upper chord, small horizontal displacements, rotations of the lower chord nodes
7	37,14	upper chord	rotations of the upper chord nodes, small horizontal and vertical displacements of the truss nodes
8	41,20	upper chord node 12	rotations of the upper chord nodes, small horizontal displacements, vertical displacement of node 12 of the upper chord
9	42,45	columns and upper chord	curvature of the upper and under chord, small horizontal displacements, large rotation at the columns
10	- 47,87	chords	rotations of the truss nodes, small horizontal and vertical displacements

Table 6.13: Results ROBOT97 without intermediate nodes

ROBOT 97, case of a structure without intermediary nodes			
Mode	α_{cr}	Remarks The buckling deformation:	Eigenvalue:
1	- 20,44	lower chord	rotations at the lower chord, no horizontal displacements, displacement of nodes on bars of the lower chord
2	- 15,64	lower chord	rotations at the lower chord, no horizontal displacements, displacement of nodes on bars of the lower chord
3	- 11,96	lower chord	rotations at the lower chord, no horizontal displacements, displacement of nodes on bars of the lower chord
4	- 8,20	lower chord	rotations at the lower chord, no horizontal displacements, displacement of nodes on bars of the lower chord
5	- 5,48	lower chord	rotations at the lower chord, no horizontal displacements, displacement of nodes on bars of the lower chord
6	- 3,25	lower chord	rotations at the lower chord, no horizontal displacements, displacement of nodes on bars of the lower chord
7	4,70	GLOBAL	rotations of the columns, horizontal and vertical displacements of the truss nodes
8	9,44	upper chord	rotations at the upper level, no horizontal displacements, displacement of node 4 of the lower chord
9	13,39	upper chord	rotations at the upper level, no horizontal displacements, displacement of node 4 and 12 of the lower chord
10	17,83	upper chord	rotations at the upper level, no horizontal displacements, displacement of node 4 and 12 of the lower chord

7. Application of the automatic procedure

The main problems, surrounding the automatic procedure, are the selection of the critical buckling coefficient α_{CR} and the way to apply sway imperfections to a structure. Methods for this were suggested in the previous part. For α_{CR}, the eigenvector corresponding to sway buckling of the structure is used. For frame imperfections, the idea is to apply to the structure the form of the buckling deformation. This is done by defining the structure with imperfections as a new initial structure in the input file, to then be able to do a non-linear analysis.

By means of two examples, the procedure will now be tested. The cases are a fixed / pin-ended element and a single frame. The main steps are as follows:
Step 1: Modelling structure
Step 2: Buckling analysis
Step 3: First or second-order analysis
Step 4: Application of frame imperfections
Step 5: Application of member imperfections
Step 6: Non-linear analysis
Step 7: Check structure to Eurocode 3

7.1 Case of a fixed / pin-ended element

Step 1: Modelling structure

Consider a fixed / pin-ended element with section IPE120 and length l = 4m. An axial load is applied at the top node, F = 250 kN. Figure 7.1 represents a computer model.

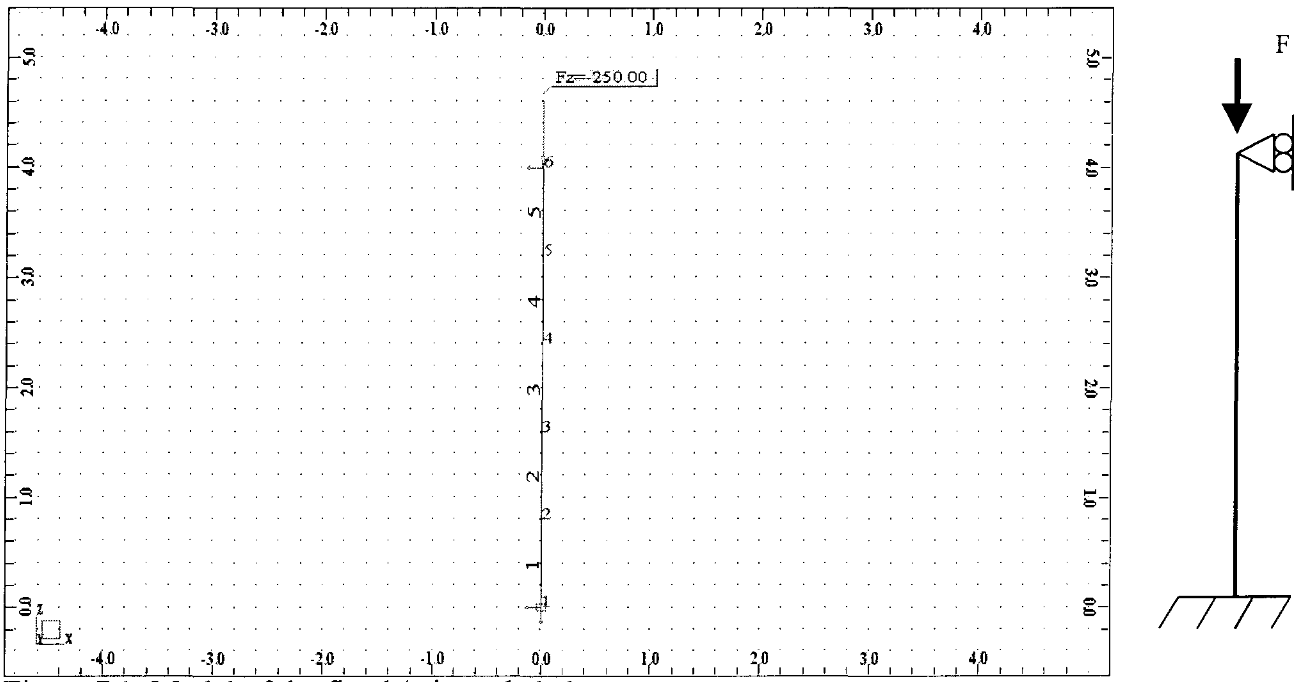


Figure 7.1: Model of the fixed / pin-ended element

Step 2: Buckling analysis

The structure is introduced in ROBOT. A linear buckling analysis of the structure is done and gives the following critical buckling coefficient and buckling length:
$\alpha_{CR} = 3.3711$
$L_f = 2.8$ m
Figure 7.2 represents the buckling deformation corresponding to $\alpha_{CR} = 3.3711$.

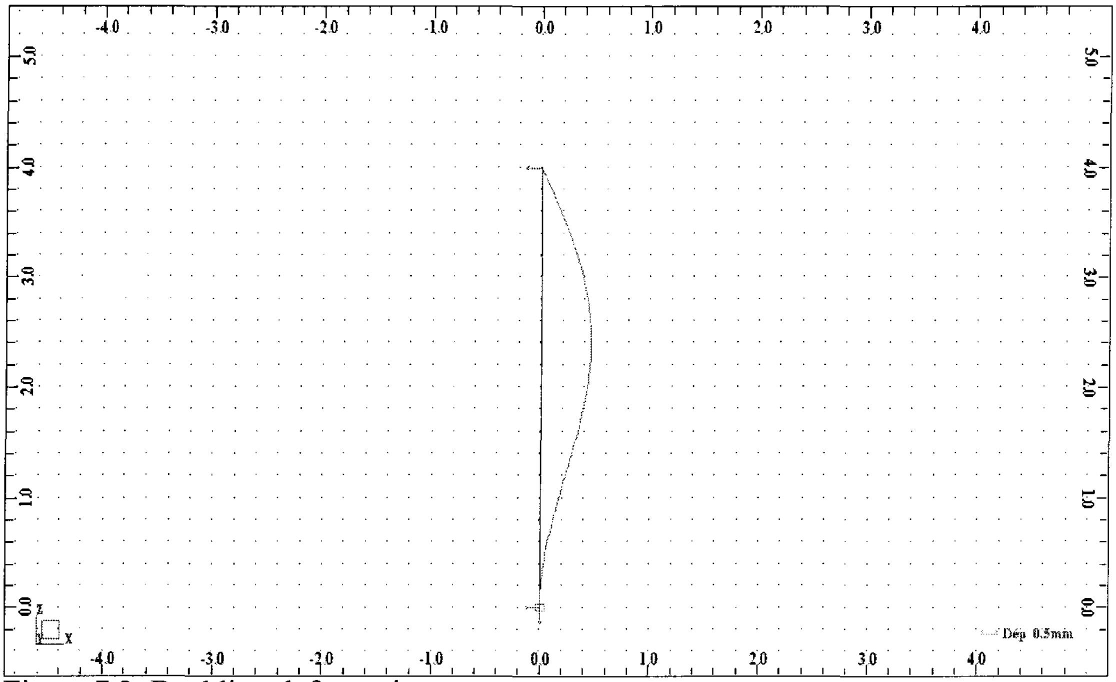


Figure 7.2: Buckling deformation

Step 3: First or second-order analysis

$\alpha_{CR} = 3.3711 < 10$ so a second-order analysis of the structure has to be done.

Step 4: Application of frame imperfections

Due to the type of structure there are no frame imperfections to be taken into account. This point needs further research.

Step 5: Application of member imperfections

Determination of the imperfection $e_{0,d}$ according to Eurocode 3:
Selection of buckling curve (Table 5.5.3, [EURO3,92])
The selected section is an IPE 120. Its thickness is: $t_f = 6.3$ mm < 40.
The section buckles about the y-y axis.
The buckling curve in that case is: buckling curve a

Class of the section (Table 5.3.1, [EURO3,92])
For IPE 120:
$d / t_w = 93 / 4.4 = 21.14 < 33$
The section is of class 1.
$\gamma_{m0} = \gamma_{m1} = 1.1$ (art. 5.5.1.3(6))

Design value of equivalent initial bow imperfection $e_{0,d}$ (Figure 5.5.1, [EURO3,92])
Eurocode 3 defines:

$$e_{0,d} = \alpha \cdot (\bar{\lambda} - 0.2) \cdot k_\gamma \cdot \frac{W_{el}}{A}$$

Where:

- $\alpha = 0.21$ (buckling curve a)
- $\bar{\lambda} = (\lambda / \lambda_1)\,[\beta_A]^{0.5} = (57.14 / 93.91) = 0.61$
  $l = 2800$ mm (step 2)
  $\lambda = l / i = 2800 / 49 = 57.14$
  $\lambda_1 = \pi\,[E/f_y]^{0.5} = \pi\,[2.1*10^5/235]^{0.5} = 93.91$
  $\beta_A = 1$ (section of class 1)

- $k_\delta = 0.23$ (Figure 5.5.1, [EURO3,92])
- $k_\gamma = (1-k_\delta) + 2\,k_\delta\,\bar{\lambda} = (1-0.23) + 2.0.23.0.61 = 1.05$
- $W_{el} = 53.10^3$ mm^3
- $A = 1320$ mm^2

$$e_{0,d} = 0.21 \cdot (0.61 - 0.2) \cdot 1.05 \cdot \frac{53 \cdot 10^3}{1320} = 3.63mm$$

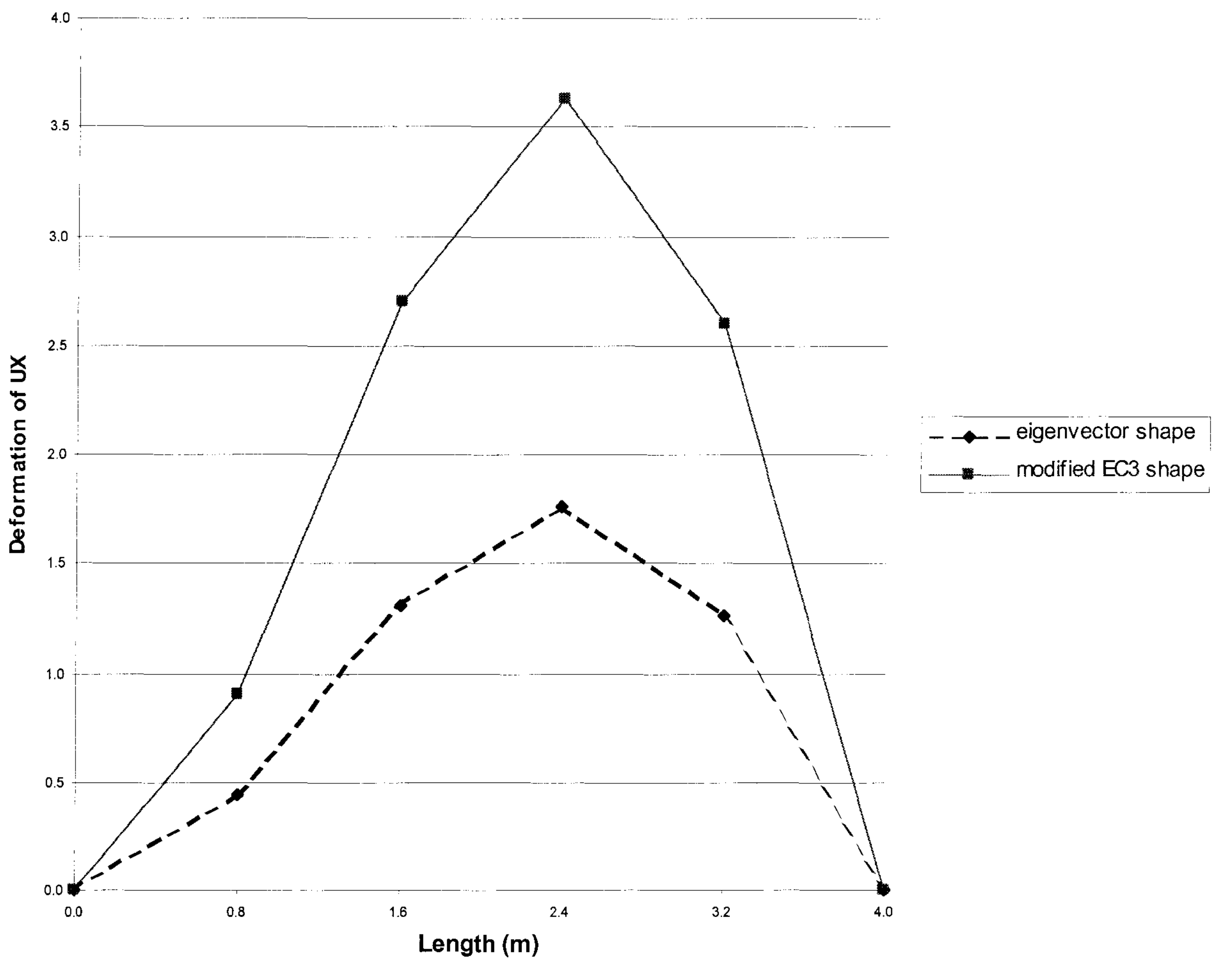


Figure 7.3: Proportional modification of the element nodes

The buckling mode and the value of the maximum deformation are known. With the help of the eigenvectors and the intermediate nodes, it is possible to determine the shape of the buckling deformation. Note that the doted line of figure 7.3 represents the eigenvectors. Now the maximum value should equal the maximum deformation defined with Eurocode 3. Proportionally the position of the different points can be determined; a modified value is obtained and is represented by the solid line of Figure 7.3.

For the buckling mode Table 7.1 represents for each node, the displacement U_X and the modified value. The modified mode "follows" the shape of the buckling mode with as maximum the one defined with Eurocode 3. It is that modified mode that will be introduced by adding it to the node positions in the input file to do a non-linear analysis of the structure.

$$U_{EC3,i} = \frac{e_{0,d} \cdot u_i}{Max(u_i)}$$

Where

$e_{0,d}$, value of the imperfection according to Eurocode 3

u_i, displacement in the buckling mode at node i

$Max(u_i)$, maximum value of the displacement in the buckling mode

Table 7.1: Proportional modification of the element nodes

Node number (i)	Node position (m)	U_i (-)	Modified $U_{EC3,i}$ (-)
1	0.0	0.00000	0.0000
2	0.8	0.00044	0.0009
3	1.6	0.00131	0.0026
4	2.4	0.00176	0.0036
5	3.2	0.00126	0.0025
6	4.0	0.00000	0.0000

<u>Step 6</u>: Non-linear analysis

A non-linear analysis with ROBOT97 has been done and gives the following results:

M_{max} = 1.16 kNm

N_{max} = 250 kN

V_{max} = 0.66 kN

<u>Step 7</u>: Check structure to Eurocode 3

Verification of the section at node 1:

Axial force and moment (art 5.4.8). The shear force is under the half of its plastic shear capacity can thus be ignored. The characteristics are:

IPE120

Section of class 1

$\gamma_{M1} = \gamma_{M0} = 1.1$ (art. 5.5.1.3(6))

$W_{pl} = 2.S_y = 2 \text{ x } 30.4.10^3 = 60.8.10^3 \text{ mm}^3$

N_{SD} = 250 kN

M_{ySd} = 1.16 kNm

<u>Axial forces capacity</u>

$N_{C,Rd} = N_{pl} = (f_y \text{ x } A) / \gamma_{M0} = (235 \text{ x } 1320) / 1.1 = 282$ kN

<u>Bending about strong axis</u>

$M_{pl,y,Rd} = (W_{pl} \text{ x } f_y) / \gamma_{M0} = (60.8.10^3 \text{ x } 235) / 1.1 = 12.99$ kNm

Axial force and bending moment about strong axis

$n = N_{SD} / N_{pl,Rd} = 250 / 282 = 0.88$

$a = (A - 2b_f t_f) / A = (1320 - 2 \times 64 \times 6.3) / 1320 = 0.39$

$M_{n,y,Rd} = M_{pl,y,Rd} (1 - n) / (1 - 0.5A) = 12.99 (1 - 0.88) / (1 - 0.5 \times 0.39) = 1.94$ kNm

1.16 < 1.94 kNm

$M_{y,Sd} < M_{n,Y,Rd}$ Condition verified

7.2 Case of a single frame

Now the case of a single frame will be treated as an example to illustrate the automatic procedure.

Step 1: Modelling structure

Consider a single frame of length 6m and height 4 m. Node 1 and 16 are pin-ended.

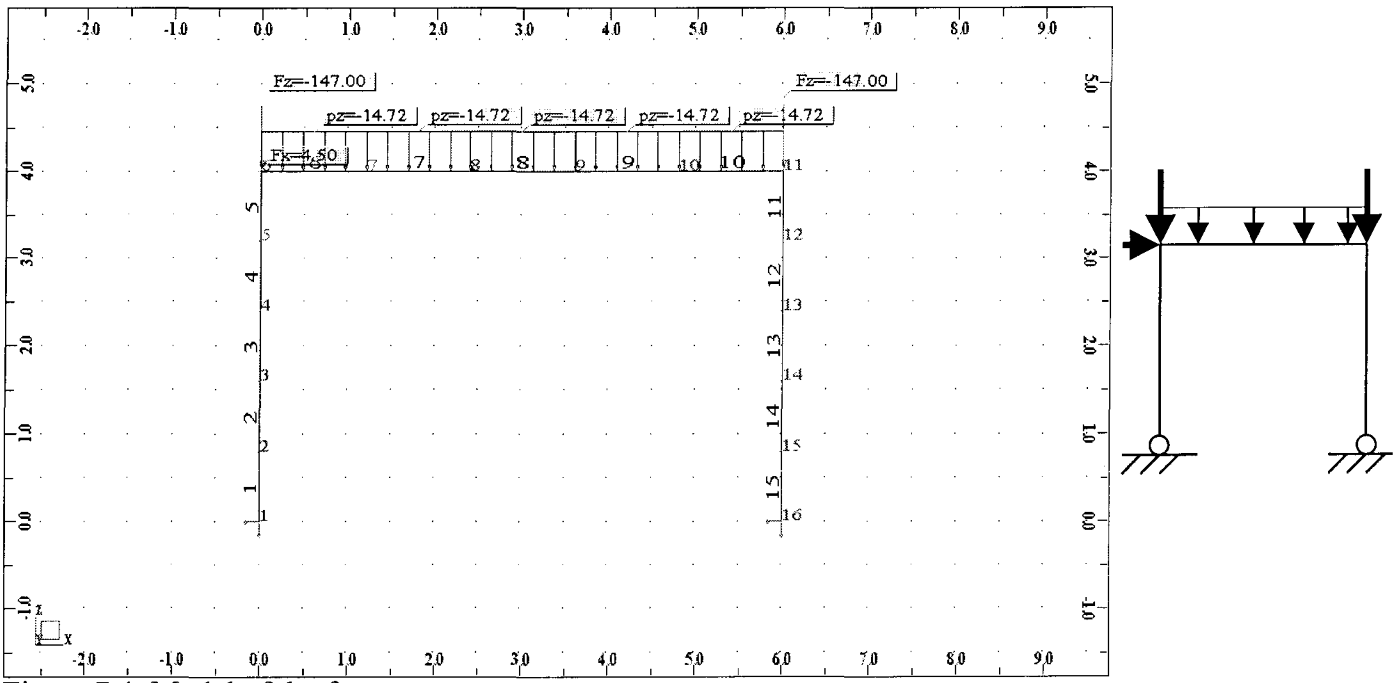


Figure 7.4: Model of the frame

The vertical elements are HEA220 and the horizontal element is an IPE120. The loading is:

- Fx = 4.5 kN at node 6;
- Fz = 147 kN at nodes 6 and 11;
- pz = 14.72 kN/m a distributed load between the elements 6 and 10.

Figure 7.4 represents a computer model.

Step 2: Buckling analysis

The structure is introduced in ROBOT. A linear buckling analysis of the structure is done and gives the following critical buckling coefficient and buckling length:
$\alpha_{CR} = 3.5118$
$L_f = 13.07$ m
Figure 7.5 represents the buckling mode corresponding to $\alpha_{CR} = 3.5118$.

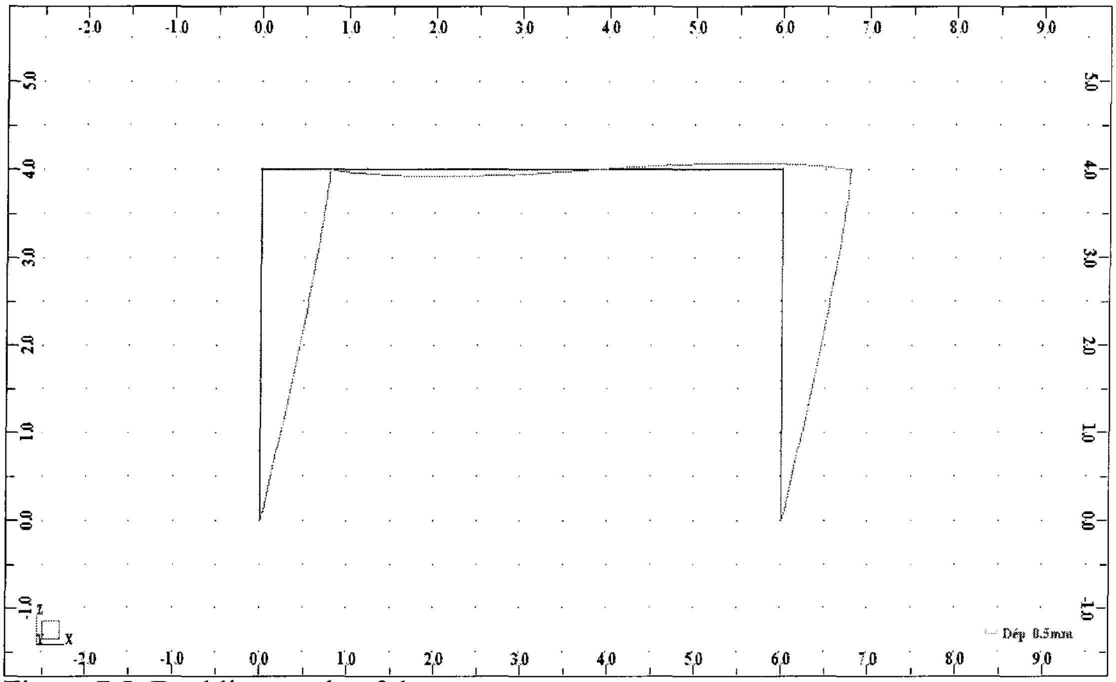


Figure 7.5: Buckling mode of the structure

Step 3: First or second-order analysis of the structure

$\alpha_{CR} = 3.5118 < 10$ so that a second-order analysis of the structure has to be done.

Step 4: Application of frame imperfections

As described before, to apply frame imperfections, the shape of the buckling mode corresponding to sway buckling of the structure will be used. The deformation is represented in Figure 7.5. Four intermediate nodes have been included to get a better accuracy of the results. Table 7.2 represents the co-ordinates of the structure, the eigenvector corresponding to sway buckling of the structure and frame imperfections. Frame imperfections are included with the help of the shape of sway buckling with the following formulas:

$$X_{new,i} = X_{old,i} + \frac{U_{X,i} \times e}{Max(U_{X,i})}$$

$$Y_{new,i} = Y_{old,i} + \frac{U_{Y,i} \times e}{Max(U_{Y,i})}$$

Where
$(X_{old,i}, Y_{old,i})$, the old co-ordinates at node i
$(X_{new,i}, Y_{new,i})$, the new co-ordinates at node i
e, the displacement used to proportionally determine the buckling shape

For the exact value of e that corresponds to the maximum horizontal deformation there is no clear definition in Eurocode 3. Therefore, it is suggested to consider the Eurocode 3's formula for the most unfavourable case, applied at the top:

$$e = \frac{L}{200} = \frac{4000}{200} = 20mm$$

Table 7.2: Modification of the co-ordinates of the structure

		Old co-ordinates		Eigenvectors		Frame imperfections	
	Node number	X (mm)	Z (mm)	UX (-)	UZ (-)	X (mm)	Z (mm)
Vert. element	1	0	0	0	0	0	0
	2	0	800	0.0008	0	5	800
	3	0	1600	0.0015	0	9	1600
	4	0	2400	0.0021	0	13	2400
	5	0	3200	0.0027	0	17	3200
	6	0	4000	0.0032	0	20	4000
Horiz. element	7	1200	4000	0.0032	-0.0003	1220	3998
	8	2400	4000	0.0032	-0.0002	2420	3999
	9	3600	4000	0.0032	0.0002	3620	4001
	10	4800	4000	0.0032	0.0003	4820	4002
Vert. element	11	6000	4000	0.0032	0	6020	4000
	12	6000	3200	0.0027	0	6017	3200
	13	6000	2400	0.0021	0	6013	2400
	14	6000	1600	0.0015	0	6009	1600
	15	6000	800	0.0008	0	6005	800
	16	6000	0	0	0	6000	0

Looking at column "frame imperfections" in Table 7.2, it is noted that the maximum horizontal deformations are between nodes 6 and 11. The X-value is the old one + 20 mm.

Step 5: Application of member imperfections

To determine member imperfections, Eurocode 3 can be used:

The vertical elements
Selection of buckling curve (Table 5.5.3 [EURO3,92])
The selected section is an HEA 220. Its thickness is: $t_f = 11$ mm < 40.
The section buckles about the y-y axis.
The buckling curve in that case is: buckling curve a

Class of the section (Table 5.3.1, [EURO3,92])
For HEA 220:
$d / t_w = 152 / 7 = 21.71 < 33$
The section is of class 1.
$\gamma_{m0} = \gamma_{m1} = 1.1$ (art. 5.5.1.3(6))

Design value of the equivalent initial bow imperfection $e_{0,d}$ (Figure 5.5.1, [EURO3,92])
Eurocode 3 defines:

$$e_{0,d} = \alpha \cdot (\bar{\lambda} - 0.2) \cdot k_\gamma \cdot \frac{W_{el}}{A}$$

Where:

- $\alpha = 0.21$ (buckling curve a)
- $\bar{\lambda} = (\lambda / \lambda_1)\,[\beta_A]^{0.5} = (139.04 / 93.91) = 1.48$
  $\lambda = l / i = 13070 / 94 = 139.04$
  $l = 13070$ mm (step 2)
  $\lambda_1 = \pi\,[E/f_y]^{0.5} = \pi\,[2.1*10^5/235]^{0.5} = 93.91$
  $\beta_A = 1$ (section of class 1)
- $k_\delta = 0.23$ (Figure 5.5.1, [EURO3,92])
- $k_\gamma = (1-k_\delta) + 2\,k_\delta\,\bar{\lambda} = (1-0.23) + 2.0.23.1.48 = 1.45$
- $W_{el} = 515.10^3$ mm^3
- $A = 6430$ mm^2

$$e_{0,d} = 0.21 \cdot (1.48 - 0.2) \cdot 1.45 \cdot \frac{515 \cdot 10^3}{6430} = 31.22mm$$

The horizontal element
Selection of buckling curve (Table 5.5.3 [EURO3,92])
The selected section is an IPE 200. Its thickness is: $t_f = 8.5$ mm < 40.
The section buckles about the y-y axis.
The buckling curve in that case is: buckling curve a

Class of the section (Table 5.3.1, [EURO3,92])
For IPE 200:
$d / t_w = 159 / 5.6 = 28.39 < 33$

The section is of class 1.
$\gamma_{m0} = \gamma_{m1} = 1.1$ (art. 5.5.1.3(6))

Design value of the equivalent initial bow imperfection $e_{0,d}$ (Figure 5.5.1, [EURO3,92])
Eurocode 3 defines:

$$e_{0,d} = \alpha \cdot \left(\bar{\lambda} - 0.2\right) \cdot k_\gamma \cdot \frac{W_{el}}{A}$$

Where:
- $\alpha = 0.21$ (buckling curve a)
- $\bar{\lambda} = (\lambda / \lambda_1)\,[\beta_A]^{0.5} = (139.04 / 93.91) = .78$
  $l = 6000$ m
  $\lambda = l / i = 6000 / 82 = 73.17$
  $\lambda_1 = \pi\,[E/f_y]^{0.5} = \pi\,[2.1*10^5/235]^{0.5} = 93.91$
  $\beta_A = 1$ (section of class 1)
- $k_\delta = 0.23$ (Figure 5.5.1, [EURO3,92])
- $k_\gamma = (1-k_\delta) + 2\,k_\delta\,\bar{\lambda} = (1-0.23) + 2.0.23.0.78 = 1.12$
- $W_{el} = 194.10^3$ mm^3
- $A = 2850$ mm^2

$$e_{0,d} = 0.21 \cdot (0.78 - 0.2) \cdot 1.12 \cdot \frac{194 \cdot 10^3}{2850} = 9.29mm$$

In the procedure description, it was noted that it was not possible to distinguish the α_{cr} of each element separately. Therefore, member imperfections are applied to the whole structure. The considered bow shape is a sinus. The imperfection is applied in the same direction as the one defined by sway buckling. The sign of the displacements in the eigenvectors gives the direction. It all comes down to adding the co-ordinates of frame and member imperfections, as represented in Table 7.3.

Table 7.3: Modified Co-ordinates of the structure

		Modified Co-ordinates of the structure			
		Frame imperfections		Frame and member imperfections	
	Node number	X (mm)	Z (mm)	X (mm)	Z (mm)
Vert. element	1	0	0	0	0
	2	5	800	23	800
	3	9	1600	39	1600
	4	13	2400	43	2400
	5	17	3200	35	3200
	6	20	4000	20	4000
Horiz. element	7	1220	3998	1220	3993
	8	2420	3999	2420	3990
	9	3620	4001	3620	3992
	10	4820	4002	4820	3996
Vert. element	11	6020	4000	6020	4000
	12	6017	3200	6035	3200
	13	6013	2400	6043	2400
	14	6009	1600	6039	1600
	15	6005	800	6023	800
	16	6000	0	6000	0

Figure 7.6 represents the modified structure, with frame and member imperfections included, to be introduced in the input file for a non-linear analysis.

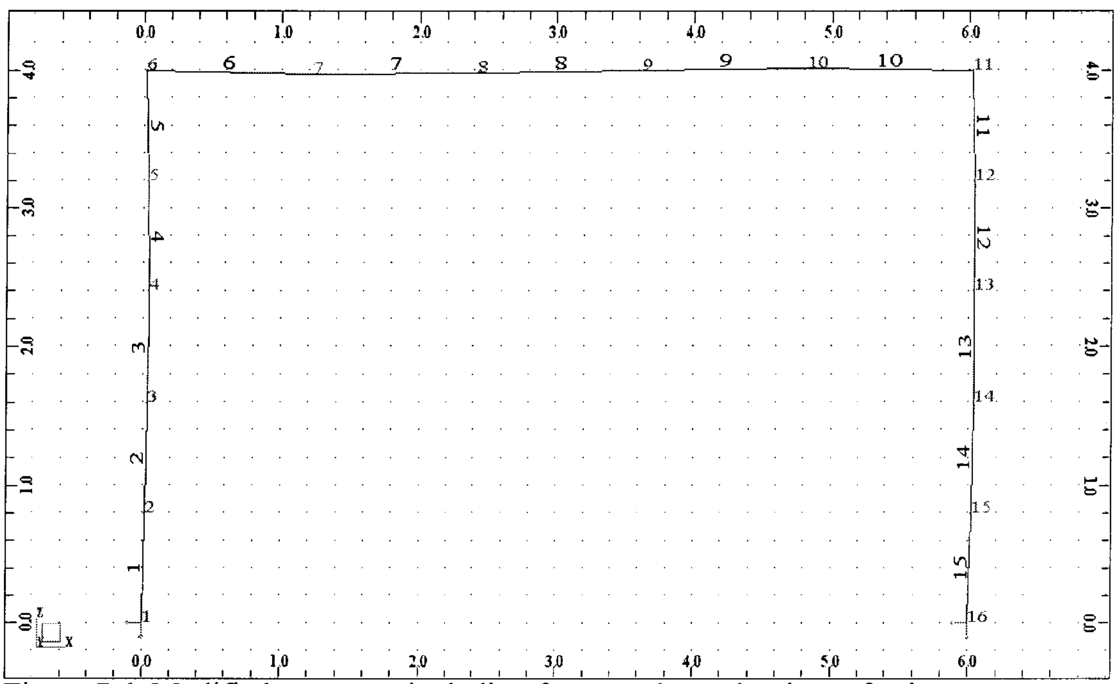


Figure 7.6: Modified structure, including frame and member imperfections

Step 6: Non-linear analysis

Table 7.4 represents forces and moments obtained after a linear and a non-linear analysis of the structure. They are placed next to the results obtained in the linear analysis.

Table 7.4: Forces and moments in the structure for a linear and non-linear analysis

		Resulting forces and moments in the structure					
		Linear analysis			Non-linear analysis		
	Node Number	FX (kN)	FZ (kN)	MY (kNm)	FX (kN)	FZ (kN)	MY (kNm)
Vert. element	1	188	-7	0.0	185	-2	0.0
	2	188	-7	-5.8	185	-4	-0.4
	3	188	-7	-11.6	185	-6	-2.2
	4	188	-7	-17.4	185	9	6.1
	5	188	-7	-23.3	185	11	12.2
	6	12	41	-29.1	11	38	-19.4
Horiz. element	7	12	24	9.7	11	20	16.0
	8	12	6	27.3	11	2	29.8
	9	12	-12	23.7	11	-15	22.0
	10	12	-30	-1.1	11	-33	-6.9
Vert. element	11	194	12	-47.1	198	8	-56.7
	12	194	12	-37.7	197	10	-50.0
	13	194	12	-28.2	197	-13	41.4
	14	194	12	-18.8	197	-16	29.9
	15	194	12	-9.4	197	-17	15.7
	16	194	12	0.0	197	-17	0.0

Step 7: Check structure to Eurocode 3

The elements are loaded by a combination of axial forces and moments (art 5.4.8). The shear force is under half of its plastic shear capacity can then be ignored.

The vertical element:
Characteristics:
HEA220
Section of class 1
$\gamma_{M1} = \gamma_{M0} = 1.1$ (art. 5.5.1.3(6))
$A = 6430\ mm^2$
$W_{pl} = 2.S_y = 2 \times 284.10^3 = 568.10^3\ mm^3$
$N_{SD} = 198$ kN
$M_{y,Sd} = 56.7$ kNm
($V << 0.5\ N_{SD}$, V is neglected)

Axial forces capacity
$N_{C,Rd} = N_{pl} = (f_y \times A) / \gamma_{M0} = (235 \times 6430) / 1.1 = 1374$ kN

Bending about strong axis
$M_{pl,y,Rd} = (W_{pl} \times f_y) / \gamma_{M0} = (568.10^3 \times 235) / 1.1 = 121.35$ kNm

Axial force and bending moment about strong axis
$n = N_{SD} / N_{pl,Rd} = 198 / 1374 = 0.14$
$a = (A - 2b_f t_f) / A = (6430 - 2 \times 220 \times 11) / 6430 = 0.25$
$M_{n,Y,Rd} = M_{pl,y,Rd} (1 - n) / (1 - 0.5a) = 121.35 (1 - 0.14) / (1 - 0.5 \times 0.25) = 119.27$ kNm

$56.7 < 119.27$ kNm
$M_{y,Sd} < M_{n,Y,Rd}$ Condition verified

The horizontal element:
Characteristics:
IPE200
Section of class 1
$\gamma_{M1} = \gamma_{M0} = 1.1$ (art. 5.5.1.3(6))
$A = 2850 \text{ mm}^2$
$W_{pl} = 2.S_y = 2 \times 110.10^3 = 220.10^3 \text{ mm}^3$
$N_{SD} = 11$ kN
$M_{Y,Sd} = 29.8$ kN
($V << 0.5 N_{SD}$, V is neglected)

Axial forces capacity
$N_{C,Rd} = N_{pl} = (fy \times A) / \gamma_{M0} = (235 \times 2850) / 1.1 = 609$ kN

Bending about strong axis
$M_{Pl,y,Rd} = (W_{pl} \times f_y) / \gamma_{M0} = (220.10^3 \times 235) / 1.1 = 47$ kNm

Axial force and bending moment about strong axis
$n = N_{SD} / N_{Pl,Rd} = 11 / 609 = 0.02$
$a = (A - 2b_f t_f) / A = (2850 - 2 \times 200 \times 8.5) / 2850 = -0.19$
$M_{N,y,RD} = M_{Pl,y,Rd} (1 - n) / (1 - 0.5a) = 47 (1 - 0.02) / (1 + 0.5 \times 0.19) = 50.98$ kNm

$29.8 < 50.98$ kNm
$M_{y,SD} < M_{n,y,Rd}$ Condition verified

7.3 Conclusion on the automatic procedure

Though some of the steps of the automatic procedure might need further research, especially the value considered for the maximum deformation for frame imperfections, the different steps work. The two previous cases show that each step can easily be computed, because each step can mathematically be defined.

8. CONCLUSIONS

The following conclusions concerning the automatic procedure can be drawn:

- The basic idea for an automatic procedure is to consider the buckling deformation as an initial imperfection; then a non-linear analysis of the structure has to be done. With ROBOT and Hercule, it is possible to include imperfections by using the eigenvector corresponding to sway buckling. The eigenvector is scaled proportionally to a prescribed imperfection value and then added to the node positions to obtain the imperfect structure to be used in a non-linear analysis.

- The following automatic procedure is suggested:

  Step 1: Modelling structure
  Make a model of the structure with four intermediate nodes per element.

  Step 2: Buckling analysis
  A linear analysis determines forces in each element of the structure. A buckling analysis gives a series of α_{CR} values corresponding to related buckling modes. Select the lowest positive α_{CR} value for sway buckling.

  Step 3: First or second-order analysis
  Compare the chosen α_{CR} to the Eurocode 3 criterion 10, to decide whether or not it is necessary to apply imperfections. Two cases can be distinguished:
    - $\alpha_{CR} > 10$, the structure is classified as non-sway. There is no need to apply imperfections; a first-order (linear) analysis of the structure is sufficient (already done in step 2). In this case, go to step 7, the strength and the stability of the structure need to be checked;
    - $\alpha_{CR} < 10$, the structure is classified as sway. Imperfections need to be taken into account. Eurocode 3 distinguishes between frame and member imperfections. A second-order (non-linear) analysis of the structure needs to be carried out, go to step 4.

  Step 4: Application of frame imperfections
  To apply frame imperfections, the shape of the buckling deformation, resulting from the buckling analysis in step 2. The eigenvector is proportionally modified to obtain the buckling shape. Eurocode 3 does not define the size of the imperfection. Therefore, it is suggested to consider L/200 as the maximum deformation, with L the "maximum height" of the structure. This imperfection value seems to be quite severe. Note that this value needs further research, but the value is used to be able to test the automatic procedure.

  Step 5: Application of member imperfections
  In the automatic procedure member imperfections need to be added to frame imperfections. The direction to apply imperfections is the one defined, using the intermediate nodes, by the sway buckling of step 2. By placing intermediate nodes, it is possible to give mathematically the directions in which each element deforms (with the eigenvector corresponding to sway buckling). The maximum deformation is defined by Eurocode 3 ($e_{0,d}$).

Step 6: Non-linear analysis
Once frame and member imperfections are applied to the structure, a non-linear analysis can be performed. Second-order forces and moments are obtained.

Step 7: Check structure to Eurocode 3
In case of a first-order analysis of the structure, use the first-order forces of step 2 to check the strength and the stability of the structure according to Eurocode 3. In case of a second-order analysis of the structure, use the second-order forces of step 6 to check the strength of the structure only.

The influence of intermediate nodes on the buckling deformation was considered and led to the following conclusion:

- The study of different structures without intermediate nodes did not prevent the programs to draw the buckling shape. The programs assume a curve between the rotations at the end nodes.

- For both programs ROBOT and Hercule, different cases showed that at least four intermediate nodes were needed to obtain good accuracy for the critical buckling value α_{CR}.

- Furthermore it was shown that adding supplementary intermediate nodes implies more buckling modes. However too low a number of intermediate nodes can lead to the situation of not having enough modes to obtain the critical buckling value corresponding to sway deformation of the structure.

The practical application of the critical buckling coefficient α_{CR} was considered:

- ROBOT and Hercule do not rank the values for α_{CR} the same way. Depending on the number of elements (the ones under tension included) in a structure, it is often necessary to regard more buckling modes to obtain the value of α_{CR} corresponding to sway buckling.

- The coefficient α_{CR} depends on the loading on the entire structure. Any load on the system next to the main structure has to be considered. For the buckling length, Eurocode 3 only takes information of the geometry into account and not the loading.

- To select the lowest buckling coefficient α_{CR}, corresponding to sway buckling of the structure, the eigenvectors are used. Sway buckling is when, for an eigenvector, the nodes other than the supports displace.

The automatic procedure, proposed in this report, was applied to an individual structural element and a portal frame but can be applied to more general structures. Cases like complex plane frames or even three-dimensional structures are possible. The automatic procedure can take three-dimensional effects into account (after a small modification of the program). However, this report is a first consideration of the problem, it describes the limits surrounding an automatic application of imperfections.

More cases need to be tested to justify the procedure. Still, there is one point that, with certainty, needs to be analysed further. The value of frame imperfections remains unclear. This report suggests considering 1/200 of the maximum height of structures. This value is however quite high. For very complex or three-dimensional structures, this suggestion is not applicable anymore, because the "maximum height" can not so easily be defined. Further research in that area needs to be done, because Eurocode 3 does not propose the necessary information.

Finally, note that the automatic procedure in this report is based on Eurocode 3. That means that all the complex steps had to be considered in that procedure. Nevertheless, the chart of figure 8.1 proposes a procedure that easily can be applied. It may not quite follow Eurocode 3, but it is always on the safe side because it always proposes to do a second-order (non-linear) analysis of structures including imperfections.

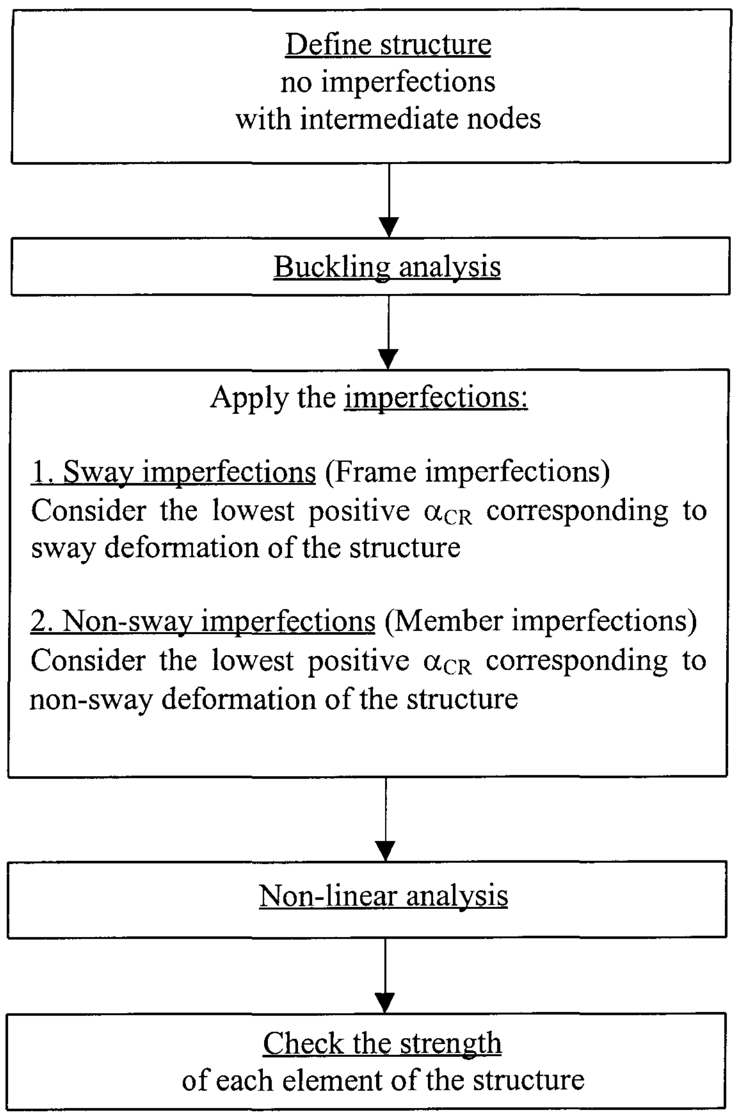


Figure 8.1: General procedure to include the effect of imperfections

BIBLIOGRAPHY

[ANSYS,97] ANSYSv5.4, procedure manuals; 1997

[CONST,96] Over spannend staal, construeren A en B; Staalbouwkundig Genootschap, 1996

[EURO3,92] Eurocode 3 "Calcul des structures en acier" et Document d'Application National – Partie 1.1: Règle générales et règles pour les bâtiments; Association française de Normalisation (afnor), Paris, 1992

[FREY,78] Frey, F., L'analyse statique non-linéaire des structures par la méthode des éléments finis et son application à la construction métallique; Thèse de doctorat de l'Université de Liège, 1978

[GALAM,76] Galambos, T.V., Guide to stability design, Criteria for metal structures; John Wiley & Sons, 1976

[HERCU,98] HERCULE, Guide de l'utilisateur; 1998

[MOREL,97] Morel, J., Calcul des structures métalliques selon l'Eurocode 3; édition Eyrolles, 1997

[MUZEAU,87]Muzeau, J.P., Modèle de l'influence d'imperfections sur la sécurité des structures métalliques en comportement non-linéaire; Thèse de doctorat de l'Université Blaise Pascal, Aubière, 1987

[NEN6770,91]NEN 6770 Staalconstructies TGB1990, Basiseisen en basisrekenregels voor overwegend statisch belaste constructies; Nederlands Normalisatie Instituut, Delft, 1991

[NEN6771,91]NEN 6771 Staalconstructies TGB1990, Stabiliteit; Nederlands Normalisatie Instituut, Delft, 1991

[PEP,95] PEPmicro Analyse plastique au second ordre de structures planes à barres, Manuel d'utilisation; Saint Rémy-lès-Chevreuse, 1995

[PETER,88] Petersen, C., Stahlbau; Friedr.Vieweg & Sohn, 88

[ROBOT,96] ROBOTv6STRUCTURES, manuel d'utilisation, volume 2; 1996

[STARK,93] Stark, J.W.B., Staalconstructies 2, dictaat faculteit Bouwkunde, nr 7276; Eindhoven University of Technology, 1993

[SNIJDER,95]Snijder, H.H., Stabiliteit (Kniklengte en Knikstabiliteit), dictaat faculteit Bouwkunde; Eindhoven University of Technology, 1995

[STRAT,77] Strating, J. e.a., Stabiliteit voor de staalconstructeur; Staalbouwkundig Genoodschap, 1977

ANNEXES

Annex A.

Utilisation et Syntaxe du Logiciel ROBOT. Exemple de Fichier dans le Cas d'une Barre Encastrée-Articulée

A.1.1 Le logiciel ROBOT97

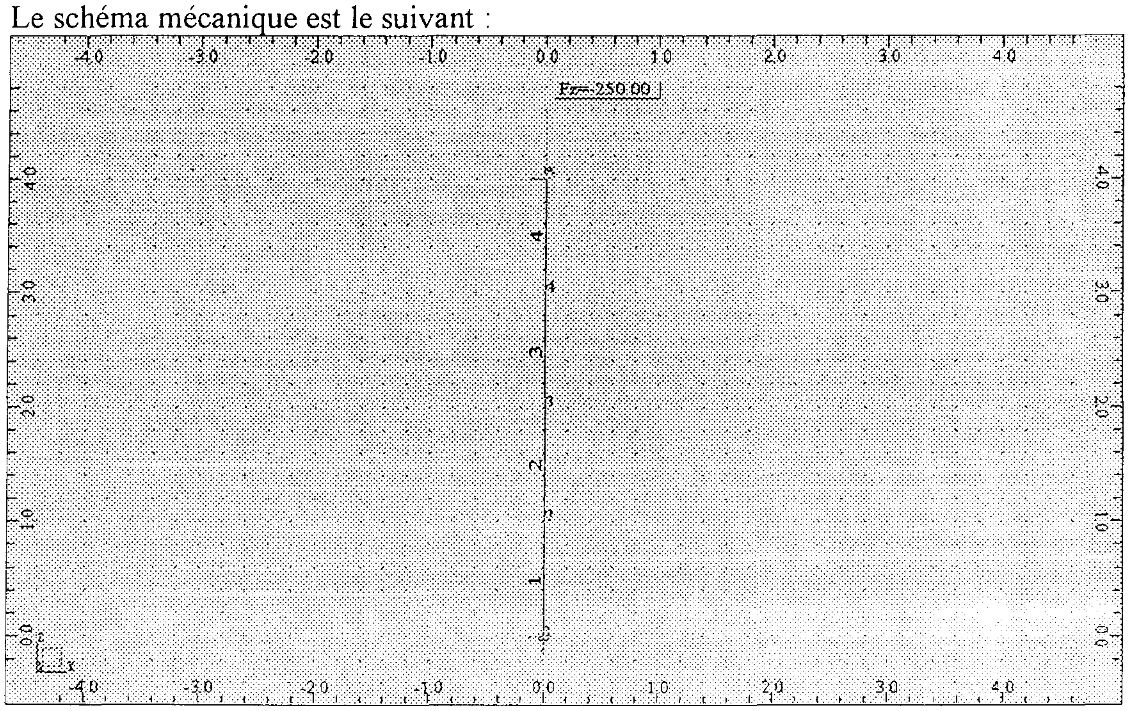
Le logiciel ROBOT fonctionne sous Windows, pour ROBOT97, et sous DOS pour ROBOTv6. Ces deux versions peuvent être utilisées de deux façons différentes. D'une part le logiciel peut être utilisé de façon interactive, à l'aide des différents menus affichés à l'écran qui peuvent être sélectionné à l'aide de la souris. D'autre part le logiciel peut être utilisé par fichier de données, dans lequel aussi bien la géométrie et le chargement de la structure sont indiqués, mais aussi dans lequel il est possible d'introduire des commandes de calculs de façon à obtenir directement tous les résultats désirés. Ces résultats sont enregistrés dans des fichiers qui peuvent être observés ou réutilisés ultérieurement. Cette méthode de fichier de données a été considérée dans la vue d'une procédure d'automatisation.

Dans ce qui va suivre, cette méthode par fichier de donnée sera plus amplement décrite.

A.1.1.aLes données d'entrée du logiciel ROBOT

Dans ROBOT97, on dispose d'un seul fichier d'entrée qui permet aussi bien de définir la structure que de prendre en compte toutes les informations de l'analyse désirée. Voici ci-après un exemple de structure suivie de son fichier d'entrée pour un cas simple :

Le schéma mécanique est le suivant :



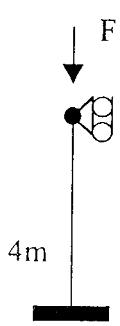


Figure A1 : cas de la barre encastrée – articulée

Dans le fichier qui va suivre, le fait d'introduire le signe « ' » permet de mettre une ligne de commentaire. La commande « FIN » arrête l'analyse du fichier.

```
' Nom de fichier: poutre     Date: 2/06/99   8:00  ROBOT 97 v. 12.5

ROBOT
PORtique PLAn
NUMérotation DIScontinue
UNItés
 Longueur=m  Force=kN  DEG


CARACTERISTIQUES
ACIER
1a4 12  SX=0.00132  IY=0.000003178              'IPE 120          'gamma 90

NOEuds
1 0 0
2 0.002 1.2
3 0.004 2.4
4 0.0024 3.2
5 0 4

ELEments
1r3 1 2

APPuis
' Libellé          - relâché
' Pas de libellé - bloqué
1
5 RZ UY

CHArgements
ANA FLA TOL 0.0001 MAXI 40 MODE 10

CAS # 1 Poutre
NOE
5 FY -250

EXE

FIN
```

Figure A2 : fichier caractéristique de ROBOT

A.1.1.b Remarques sur les fichiers d'entrées :

Dans le fichier de la figure A2, des remarques peuvent être faites quant à la syntaxe utilisée.

A.1.1.b1 Syntaxe concernant les caractéristiques des éléments.

Dans ROBOT, les caractéristiques des éléments peuvent être définis de deux manières différentes. Une méthode consiste à utiliser le catalogue des profils de ROBOT. Par exemple on peut utiliser la commande:

12 IPE120
Les caractéristiques du profil IPE120 du catalogue de ROBOT seront pris en compte pour l'analyse de la structure, et appliquées à la barre numéro 12.

Dans le cadre de cette étude, une analyse comparative est faite avec le logiciel Hercule. Pour éviter les erreurs de précision dus à la définition des profils, il est dans ROBOT aussi possible de définir les caractéristiques "une à une". Par exemple le cas précédent peut s'écrire:

12 SX=0.00132 IY=0.000003178

Il est possible, dans ROBOT, de n'indiquer que les caractéristiques utiles à l'analyse de la structure. Dans le cas de ci-dessus, les caractéristiques sont la section et le moment d'inertie de flexion (ici dans l'axe faible).

A.1.1.b2 Syntaxe concernant les nœuds

On dispose d'une commande pour la disposition des nœuds intermédiaires. Une fois les coordonnées des nœuds extrêmes fournies, cette commande numérote automatiquement et génère, par des intervalles constants, des nœuds intermédiaires.

Les nœuds de l'exemple précédent peuvent être générés par répétition comme suit :

1R4 0 0 0 4
<numéro nœud> Répéter <r> <x1><y1><x2><y2>
(x1,y1) correspond au premier nœud de la liste et (x2,y2) correspond au dernier nœud de la liste.

Mais si des imperfections doivent être générés, les nœuds intermédiaires alors déformés ne seront plus alignés et cette commande ne sera alors plus valable.
Dans ROBOT, pour générer les imperfections géométriques, il est recommandé de définir « manuellement » les coordonnées de chaque nœud intermédiaire.

A.1.1.b2 Syntaxe concernant les éléments

La définition des éléments peut se faire de la même manière que la définition des nœuds :

- élément par élément :

1 1 3
3 3 5
cette fonction définie l'élément trois entre les nœuds 3 et 5

- par répétition (équivalent à la précédente):

  1R1 1p2 3p2
  <n°élément> Répéter <r> <n°nœud origine> Pas <pas> <n°nœud fin même élé> Pas <pas>
  Le paramètre <pas> permet de considérer des éléments dont la numérotation des nœuds n'est pas forcement consécutive.

A.1.1.b3 Syntaxe concernant les analyses de flambements

Les différentes analyses de flambement permettent de déterminer les coefficients α_{CR} des différents modes de flambement. De plus, grâce à ces analyses, il est possible d'obtenir les différents modes propres et vecteurs propres de flambement permettant d'afficher *une allure* de la déformée de flambement. Finalement, les longueurs de flambement de chaque barre calculées en fonction du coefficient d'éloignement de la charge critique donnent souvent pour certaines barres des valeurs erronées .

Trois cas peuvent se distinguent dans ROBOT :
- analyse de flambement en linéaire
- analyse de flambement en non-linéaire (petit déplacement)
- analyse de flambement en non-linéaire P-DELTA (grand déplacement)

Les commandes de ces trois cas sont les suivantes :
- ANAlyse **FLA**mbement **TOL**érance=<t> **MAX**itération=<limite> **MOD**es=<n>
- ANAlyse **NON**linéaire **FLA**mbement **TOL**érance=<t> **MAX**itération=<limite> **MOD**es=<n>
- ANAlyse **PDE**lta **FLA**mbement **TOL**érance=<t> **MAX**itération=<limite> **MOD**es=<n>

Dans les commandes de ci-dessus, et dans le fichier de données de façon général, seuls les trois premières lettres sont obligatoires.

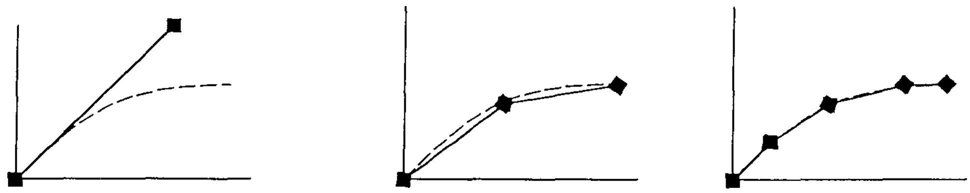
Description des commandes qui se rattachent à ces analyses :
1) **FLA**mbement active le calcul de flambement pour la recherche du α_{CR}

2) **NON**linéaire active le calcul non-linéaire

3) **PDE**lta active le calcul non-linéaire en grand déplacement

4) **TOL**érance permet de définir la précision désirée. **TOL = 0.0001**
   La tolérance permet donc de voir si la structure converge lors des différentes itérations. Pour une tolérance très faible une précision beaucoup plus fine sera obtenue, mais ceci n'est d'une part pas nécessaire dans notre cas, et d'autre part cela rallonge grandement la durée nécessaire des calculs. Une tolérance de 0.0001 est suffisante.

5) **MAX**itération définit le nombre d'itérations au bout desquelles s'arrête la session de calcul et s'affichent les résultats si la tolérance prédéfinie n'est pas encore atteinte.
Cette commande offre la possibilité d'affiner l'approche du comportement non-linéaire.



1 itération 2 itérations 3 itérations

Figure A3 : Précision de la non linéarité par la variation du nombre d'itérations

Il est possible suivant le cas de complexité de la structure, de faire varier le nombre d'itération. Pour cette étude a été choisi de considérer au moins quarante itérations, pour pouvoir obtenir des résultats intéressants. MAXI = 40. Ce point sera repris dans les différents exemples de cette partie.

6) **MOD**es définit le nombre de modes de flambement. **MOD = 10** (max)
Souvent pour les éléments simples du style barre articulée – encastrée sans nœuds intermédiaires, seuls les nœuds extrêmes sont définis. La résolution matricielle, du fait du manque de points de calculs, ne peut déterminer que les vecteurs propres solutions des nœuds extrêmes. Et par conséquent, seul *un* mode de flambement est obtenu.
Théoriquement le nombre de mode correspond au nombre maximal du nombre de lignes de la matrice de rigidité. Ce qui a donc à voir avec le nombre de degré de liberté des différents nœuds. Plus la structure comporte des nœuds, plus des informations peuvent être données sur les degrés de liberté de ces points et donc plus le nombres de mode demandé peut être grand.

En pratique du fait du grand nombre de calcul que cela engendre de demander plus de modes, le logiciel robot semble saturer. Les calculs deviennent alors très lents, et même une fois terminés la lecture des résultats est très lente aussi.

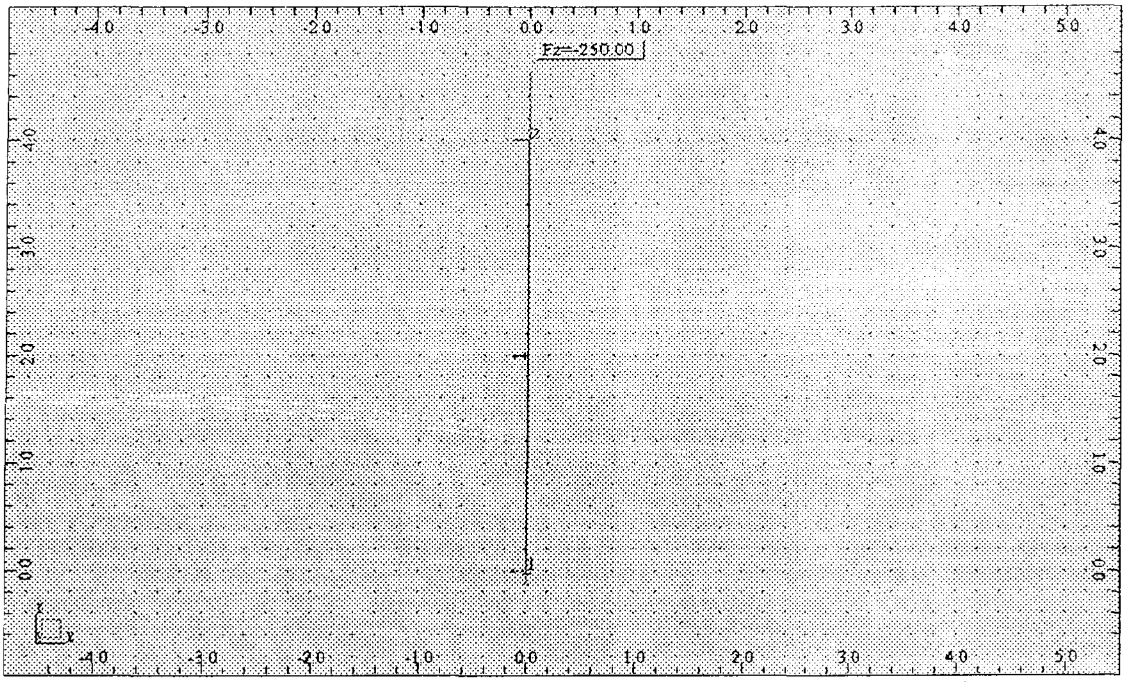


Figure A4 : Exemple de poutre encastrée articulée, sans nœuds intermédiaires.

Tableau A1: Résultat ROBOT, après une analyse modale avec 10 modes.

Barre	Cas	Longueur de flambement Y (m)	F.Critique (kN)	Coef.crit.	Précision
1	1	2.2943	1251.15	5.00E+00	1.58E-16

Donc, dans un cas plus général, il semblerait que plus il y a de nœuds intermédiaires plus le nombre de modes demandé peut-être important.
Les différentes valeurs du coefficient α_{CR} sont classées par ordre croissant, de ce fait il n'est pas nécessaire de demander un trop grand nombre de modes non plus. Dans la plupart des cas testés, dix modes étaient suffisants pour voir comment la structure se comportait graphiquement. De plus, pour ces différents modes, la valeur de ce coefficient d'amplification de charge pouvait croître très fortement et par conséquent ne plus être révélateur.

Dans ROBOT, si pour une structure après analyse le nombre de modes fournis ne peut pas être atteint (ici 10 modes), alors le logiciel l'indiquera et calculera (si on le désire) la structure suivant le nombre de modes qu'il pourra. Par exemple le cas de ci-dessus pour 10 modes indiqués seul 1 mode a pu être déterminé.

A.1.1.b4 Syntaxe concernant les sorties calculs

Il est possible, dans le fichier de données de ROBOT, d'une part d'exécuter automatiquement l'analyse de la structure et d'autre part de sélectionner les résultats.

```
....
EXE

SORtie

FIChier "nom du fichier1" format ascii
RESultats
FLAmbement
        COEfficient (CRItique)  FORce (CRItique)  LONgueur VECteurs (PROpres)
        NOEUds  <liste_noeuds >
        ELEments <liste_éléments>
        MODes   <liste_modes >

FIChier "nom du fichier2" format ascii
RESultats
EFForts
        ELEments <liste_éléments> ) _
        EXTrêmes [ LOCaux | [ GLObaux  ( EXClusif )  ] ) _

FIN
```

Figure A5: Exemple de la fin d'un fichier de données

La commande EXE permet de lancer l'analyse définie dans le fichier de données. Dans le cas où un ou plusieurs fichiers résultats veulent être créés, la commande SORtie est utilisée. Celle-ci est suivie de la commande FIChier pour laquelle le nom et le format des fichiers résultats sont indiqués.

Le format peut avoir les formes suivantes:
- Format ASCII
- Format Excel
- Format 123

Ce format dépendra du type de procédure d'automatisation considérer.

Dans la figure A5, deux fichiers sont créés. Le premier fichier correspond aux résultats de l'analyse de flambement. Le second fichier regroupe les différents efforts de la structure.

Dans le cadre d'une procédure d'automatisation, le fait d'introduire des fichiers résultat est une étape importante, car à la base de ces résultats il sera, par exemple, possible après l'analyse des différents coefficients d'éloignement de la charge critique de classer la structure.

A.1.1.c Les résultats.

Dans le cas d'une analyse de flambement une fois l'analyse lancée, les résultats peuvent être demandés de façon manuelle à l'aide de la souris ou bien sous la forme de fichiers résultats.

Différents résultats peuvent être demandés. Il est possible lors de l'écriture du fichier de données de sélectionner les résultats. Cette sélection peut toucher aussi bien la géométrie (une barre, un nœud, ...) que les résultats (efforts, déplacements, coefficients critiques, ...).

Les résultats suivants peuvent être demandés :

1) **Le coefficient d'éloignement de la charge critique** α_{CR}, étudié plus dans les détails ultérieurement.

2) **Les charges critiques**, la charge critique d'Euler, correspond à la multiplication de ce coefficient α_{CR} par la force axiale présente dans la barre considérée :

   $P_{CR} = \alpha_{CR}.P$

3) **Les longueurs de flambement L_f** de ces barres, est obtenue à l'aide de la formule théorique d'Euler :

   $$L_f = \sqrt{\frac{EI}{P_{CR}}}$$

   avec
   E, le module de Young
   I, le moment d'inertie de flexion de la barre

4) **Les élancements λ** :

   $$\lambda = \frac{L_f}{i}$$

   $$i = \sqrt{\frac{I}{A}}$$

   avec A, la section de la barre.

Voici un exemple de tableau résultat de ROBOT, dans le cas d'une barre encastrée - articulée avec trois nœuds intermédiaires, et dont 3 modes de flambements ont été demandés.

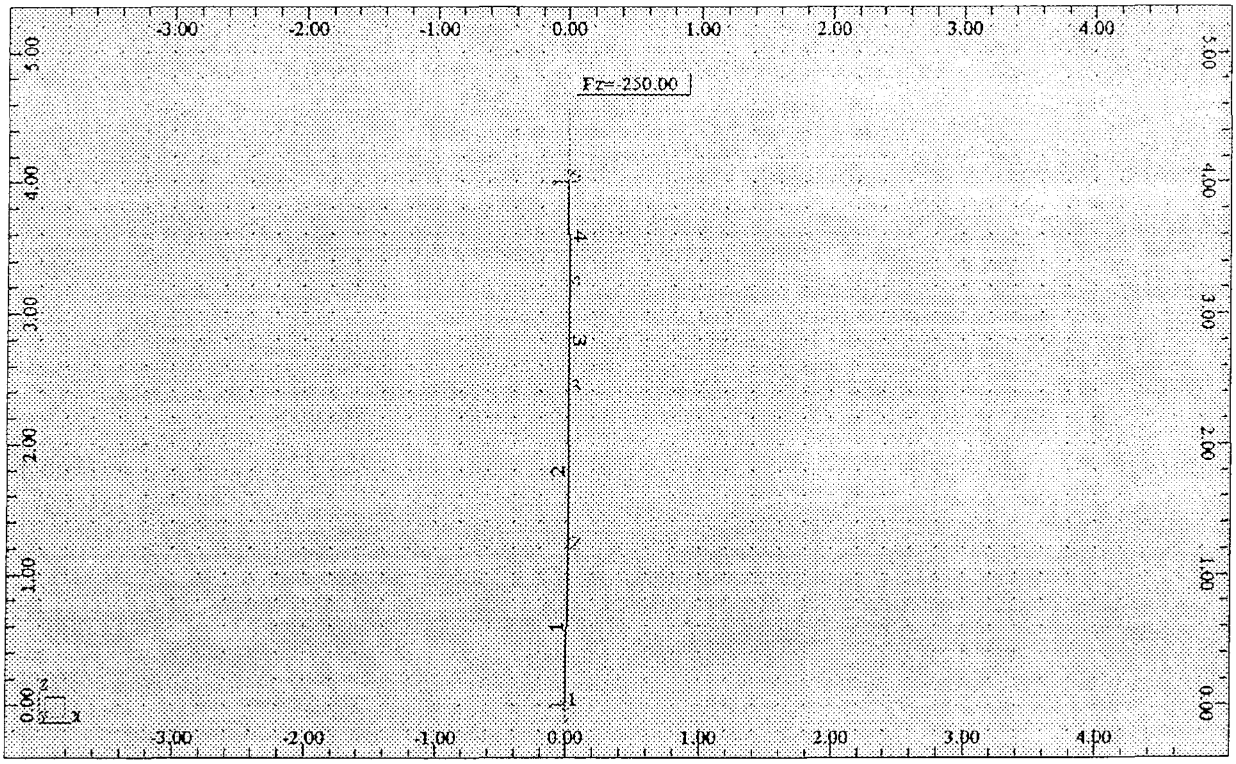


Figure A6: cas de la poutre articulée encastrée, avec 3 nœuds intermédiaires.

Tableau A2 : exemple des résultats sous ROBOT97 pour le cas de la barre encastrée - articulée

Barre	Cas	Mode	Lf - Y (m)	Lf - Z (m)	λ-Y	λ-Z	Fcr (kN)	αcr	Précision
1	1	1	2.8	(N/A)	57.0	(N/A)	844	3.38	4.89E-16
2	1	1	2.8	(N/A)	57.0	(N/A)	844	3.38	4.89E-16
3	1	1	2.8	(N/A)	57.0	(N/A)	844	3.38	4.89E-16
4	1	1	2.8	(N/A)	57.0	(N/A)	844	3.38	4.89E-16
1	1	2	1.6	(N/A)	32.9	(N/A)	2529	10.12	1.54E-13
2	1	2	1.6	(N/A)	32.9	(N/A)	2529	10.12	1.54E-13
3	1	2	1.6	(N/A)	32.9	(N/A)	2529	10.12	1.54E-13
4	1	2	1.6	(N/A)	32.9	(N/A)	2529	10.12	1.54E-13
1	1	3	1.1	(N/A)	22.9	(N/A)	5208	20.83	3.65E-13
2	1	3	1.1	(N/A)	22.9	(N/A)	5208	20.83	3.65E-13
3	1	3	1.1	(N/A)	22.9	(N/A)	5208	20.83	3.65E-13
4	1	3	1.1	(N/A)	22.9	(N/A)	5208	20.83	3.65E-13

Dans le tableau X, les remarques suivantes peuvent être faites :

- Le fichier de données indique bien une analyse dans le plan. Ceci explique pourquoi pour ce cas aucune longueur de flambement et d'élancement ne sont pas indiqués dans le tableau.
- Bien que la barre soit discrétisée, la longueur de flambement indiquée pour chaque élément correspond bien à celle de la barre entre les deux appuis.
  Un calcul manuel théorique, à l'aide de la formule d'Euler montre que L_f = 2.8 m. Ce résultat est proche de celui de ce tableau en mode 1, en effet L_f = 2.7932 m. Il en est de même pour le coefficient d'éloignement de la charge critique α_{CR} = 3.36 (Euler) pour α_{CR} = 3.38 (ROBOT).

- La valeur du coefficient critique déterminé par mode est donnée pour toutes les barres. Il n'est pas possible de dire dans le tableau ci-dessus si l'on est dans le cas d'un flambement local ou d'ensemble, rien qu'en observant les valeurs du coefficient d'éloignement de la charge critique.

5) **Les vecteurs propres** permettent d'obtenir l'allure des déformations des différents modes de flambement (voir aspect bibliographique). L'utilisateur sait alors s'il s'agit de l'instabilité d'une barre isolée, d'un ensemble de barres ou d'une sous structure. Ce point sera amplement décrit dans les exemples de cette partie, notamment au niveau de l'analyse des résultats. Reprenons le cas de la poutre articulée encastré. Voici ci-dessous les différents vecteurs correspondant aux différents modes de flambement (trois modes dans cet exemple)

Tableau A3 : valeurs des vecteurs propres correspondant aux différents modes de flambement.

Noeud	Cas	Mode	Vect. propre UX	Vecteur propre UZ	Vecteur propre RY
1	1	1	0	0	0
1	1	2	0	0	0
1	1	3	0	0	0
2	1	1	0.0009	0	0.0011
2	1	2	0.0011	0	0.0007
2	1	3	-0.0008	0	0.0004
3	1	1	0.0018	0	0
3	1	2	0.0003	0	-0.0016
3	1	3	0.0003	0	-0.0004
4	1	1	0.0013	0	-0.0012
4	1	2	-0.0006	0	-0.0002
4	1	3	-0.0006	0	-0.0007
5	1	1	0	0	-0.0018
5	1	2	0	0	0.0012
5	1	3	0	0	0.0016

Sans trop entrer dans les détails au niveau de ces résultats, dans le plan xOz, le vecteur propre se définit comme suit :

$$\vec{u} = (Ux, Uz, Ry)$$

Dans le cadre d'une procédure d'automatisation, pour des structures définies dans l'espace, la méthode qui sera utilisée dans cette partie sera valable. Le vecteur propre considéré sera alors pris dans l'espace de la façon suivante :

$$\vec{u} = (Ux, Uy, Uz, Rx, Ry, Rz)$$

6) **Les résultats classiques** peuvent être obtenus pour la vérification des structures. Il s'agit des Réactions, Déplacements, Efforts et Contraintes qui sont données pour un chargement correspondant à $\alpha = 1$

Les résultats peuvent être sélectionnés et filtrés suivant l'étude désirée. Mais par défaut, les résultats sont de la forme suivante:

Barre	Noeud	Cas	FX (kN)	FZ (kN)	MY (kNm)
1	1	1	250	0.59	-0.71
1	2	1	250	0.59	0
2	2	1	250	0.59	0
2	3	1	250	0.59	0.72
3	3	1	250	0.32	-0.72
3	4	1	250	0.32	-0.46
4	4	1	250	0.57	-0.46
4	5	1	250	0.57	0

On constate dans le tableau de ci-dessus que les résultats sont données pour les nœuds introduits dans la structure. Dans le cas de dimensionnement d'une structure ceci peut ne pas être suffisant. Il est très courant que les efforts maximaux ne soient pas aux environs des nœuds définis. Le logiciel ROBOT propose la fonction
EFForts **EXT**rêmes (**LOC**aux ou **GLO**baux), elle permet par barre de considérer son effort maxi.

Une méthode graphique peut aussi être utilisée pour retrouver les efforts maxi de la barre et la position de ce maxi sur la barre. Dans le menu principal, dans "Résultats" → "Analyse détaillée"→"Points de division"→"Re-générer". Après cette série de commande un tableau apparaît sur l'écran dans lequel, la valeur du moment maxi est donnée ainsi que sa position sur la barre.

Voici donc comment il est possible d'utiliser ROBOT pour pouvoir faire une analyse de flambement pour une structure générale et obtenir les résultats nécessaires à la procédure d'automatisation.

Dans ce qui va suivre nous allons tout d'abord décrire le logiciel HERCULE, bien que les principes d'utilisations sont proches. Et par la suite nous allons considérer différents cas pour pouvoir rappeler et montrer les différents problèmes poser dans cette description et de celle qui va venir.

Annex B.

Utilisation et Syntaxe du Logiciel Hercule. Exemple de Fichier dans le Cas d'une Barre Encastrée-Articulée

B.1.2 Le logiciel HERCULEv33.01

Le logiciel d'Hercule, contrairement à ROBOT ne peut s'utiliser que sous DOS. Son utilisation est bien différente de ROBOT, car Hercule ne fonctionne pas de façon interactive. Pour ce logiciel, les différentes données sont réparties dans plusieurs fichiers. Le nombre de fichiers dépend du type d'analyse et des résultats que l'on désire obtenir. Il est possible de faire des vérifications sur écran de certaines étapes importantes (vérification graphique de la structure, dessins des différents modes de flambement et tableaux des résultats). Mais il n'y a pas pour l'instant d'autre méthode que de créer différents fichiers de données appropriés.

Dans ce qui va suivre, la méthode d'utilisation « générale » sera décrite, surtout au niveau de la syntaxe des commandes principales. Puis quelques fichiers types et quelques exemples seront décrits (sorties de tableaux, graphiques,...).

B.1.2.a Procédure d'utilisation et syntaxe du logiciel Hercule

Pour pouvoir comprendre ce logiciel, un certain nombre de manœuvres doivent être décrites.

1) Tout d'abord, il faut effacer tous les fichiers qui ont été créés lors des analyses précédentes. Taper :

DEL *.h??
DEL *.0??

2) Ensuite il faut initialiser l'affaire. par exemple :

H0 poutr/INIT :1

H0 permet d'initialiser l'affaire « poutr »
:1 permet de revenir directement sous dos pour pouvoir lancer les autres modules

3) Il faut écrire ses fichiers pour les modules qui correspondent à son type d'analyse.
HERCULE regroupe 9 modules différents. Chacun de ces modules à une fonction particulière, il y a par exemple un module pour la définition de la structure et son chargement (module 1), un module de résultats (module 4), etc.
Pour pouvoir lancer ces modules, des fichiers de données indépendants doivent être préparés à l'aide d'un éditeur de texte. Le nom du fichier doit respecter certaines règles, doit être de la forme :

xxxxxy.dhn
avec
xxxxx, nom de l'affaire
y, caractère facultatif
.d, 2 caractères imposés signifiant Données Hercule
hn, nom du module utilisant ces données

Par exemple : poutr.dh1, données du module 1 de l'affaire poutr

Pour pouvoir exécuter ces différents modules, une fois l'affaire initialisée, on doit utiliser la commande de la forme suivante :

C:\HERCULE>hn xxxxxy exécution du module hn pour l'affaire xxxxx avec le fichier de données xxxxxy.dhn

Les différents modules sont :

- Le module H0 : initialisation – Suivi de l'affaire
- Le module H1 : Mise en forme des données
- Le module H2 : Définition et résolution d'un problème
- Le module H4 : Calcul et impression des résultats
- Le module H5 : Vérification – Dessin
- Le module H6 : Calcul des modes propres de vibration ou de flambement
- Le module H7 : Calcul des réponses aux excitations dynamiques
- Le module H9 : Super structures
- Le module HA : intégration pas à pas dans le temps des équations de mouvement

4) Les fichiers de sorties générés par Hercule doivent être étudiés, pour distinguer des erreurs éventuelles ou rechercher les résultats.

Après l'exécution de chaque module, des fichiers destinés à l'utilisateur sont générés. On distingue deux types :

- sorties destinées à l'utilisateur :

xxxxx**m**.ppp fichier des messages
xxxxx**r**.ppp fichier des résultats
xxxxx**d**.ppp fichier de dessin
avec
xxxxx, nom de l'affaire
ppp, numéro du pas (3 chiffres)
exemple :
poutrm.003 message du troisième passage pour l'affaire poutr

- fichiers de travail :

Ces fichiers sont utilisés pour conserver les informations des différents passages de chaque module. Hercule gère totalement ces valeurs. Ils sont de la forme :
xxxxx.hnn ou xxxxx.nnl
avec
xxxxx, nom de l'affaire
nn, numéro du fichier (2 chiffres)
l, label du fichier (0 ou 1 lettre)
exemple :
poutr.h20 poutr.46a

B.1.1.b Fichier de données pour le logiciel Hercule pour une analyse de flambement.

Nous allons maintenant d'une part décrire les différentes étapes d'Hercule pour une analyse de flambement. Et d'autre part montrer un exemple des fichiers de données correspondant à ces différentes étapes.

1) Lignes de commandes pour l'analyse de flambement

Dans le cadre de l'automatisation, un fichier.bat équivalent peut être écrit :

```
cd c:\hercule\bin
del *.h??
del *.0??
h0 poutr/init :1
h1 poutr.dh1   (fichier de données utilisé par le module H1)
h2 poutr.dh2   (fichier de définition de calcul (un calcul statique))
h2 poutr2.dh2 (2e passage, fichier de définition de calcul (un calcul de flambement))
h6 poutr.dh6   (fichier de calcul des modes de flambement)
h4 poutr2.dh4 (sortie des résultats)
h5 poutr.dh5   (fichier de vérification (dessins des positions des nœuds))
h5 poutr2.dh5 (fichier de vérification (dessins des différents modes de flambement))
```

2) Fichiers d'entrée

Remarques :

- Dans les différents fichiers, le fait d'apporter le signe suivant « / » permet de placer des remarques.
- Comme pour le logiciel ROBOT, seul les trois premières lettres sont obligatoires.
- Après chaque fichier, bien introduire le mot « FIN » suivi de « ↵ » sinon le module indiquera une constante anormalité.

```
/Logiciel HERCULE
**barre articulée-encastrée avec 3 nœuds intermédiaires**
/
/ unité m et kN
NUM IMP ELE

/Caractéristiques du matériau
/acier
/syntaxe: n°mat E      nu   G        coeff.dil  poids_spéc.
CAR MAT   1     210E6 0.3 80.77e6 11E-6      78.5

/TYPe BARre n°mat S Izz Iyy Iyz J S1y S1z μ
TYP  BAR  1  1  0.00132  0.00000318  0.00000277  0  0.00000000171
0.000528 0.000806 0 /ipe120

/ numérotation des noeuds
GROupe NOEud 1
/syntaxe/LISte NOEud n°repère n°noeud   x     y     z
     LIS NOE 0 1  0        0 0
     LIS NOE 0 2  0.002  0 1.2
     LIS NOE 0 3  0.004  0 2.4
     LIS NOE 0 4  0.0024 0 3.2
     LIS NOE 0 5  0        0 4

/ liste des éléments
/GROUpe ELEment 1 à 2 NOEuds
     GRO ELE  1 2
     LIS ELE ELE 1 1,,5

/ description des liaisons externes
     GRO LIA 1
     APP SIM x y z rx ry rz 1      / encastrement
     APP SIM x y rz rx 5           / articulation

/ description des groupes de charges
     GROupe CHArge 1 / Poids propre
     POIds PROpre Z TOUs

     GRO CHA 2 / Charge du portique
     POIds PROpre Z TOUs
     CHArge NOEud GLObal Z -250 5
FIN
```

Fichier de données pour le module H1

Remarque sur les fichiers du module H1 quant à la syntaxe des nœuds :
Comme dans ROBOT, il est possible de générer automatiquement des nœuds intermédiaires. Et une fois encore, cette génération de nœuds se fait le long d'une droite. Par contre, le logiciel Hercule offre la possibilité, lorsque les barres sont composées de plus d'un nœud intermédiaire, de déplacer automatiquement ces nœuds suivant un arc de sinusoïde ou de parabole, ceci par la commande :

DEPlacer NOEuds ARC {SINus ou PARabole} e_0 $n_1,\ldots n_n$
Avec
e_0, la flèche au centre de l'arc
$n_1,\ldots n_n$, liste des numéros des nœuds

Bien que cette commande impose une certaine discrétisation de la structure, elle est très facile d'application.

Le fichier suivant permet d'activer la structure. Hercule, lors de ce module, détermine les différentes matrices de rigidité, et calcul selon les différents cas de charges l'énergie potentielle.

```
/ Module H2    (calcul)
/
 barre articulée - encastrée
ACTiver ELEments TOUs
ACTiver LIAisons TOUs
ELI COM Y RX RZ TOU / à mettre absolument pour obtenir le
                      flambement dans le plan de la
                      structure
CAS CHArge 1
 POIDS
 CHA APP G1
CAS CHA 2
 FORCE
 CHA APP G2         / même cas que celui du module 1
FIN
```

Fichier de données pour le module H2, premier passage

Remarque sur les fichiers du module H2 dans le cas d'une étude dans le plan :
Par défaut, le logiciel Hercule travail en trois dimensions, donc pour obtenir les résultats dans le plan de la structure, il est nécessaire d'ajouter la commande qui permette d'éliminer toutes les composantes des autres plans. Cette commande est la suivante :

ELIminer COMposante {X Y Z} {RX RY RZ} $n_1,\ldots n_n$

Le fichier suivant, qui est en fait un deuxième passage du module H2, permet de générer les données nécessaires à l'analyse de flambement.

```
/
/ Module H2    (calcul)
/
* barre articulée - encastrée *

/
CAL FLA /GRA DEP
ETA INI CAS 2
ACTiver ELEments TOU
ACTiver LIAisons TOU
ELI COM Y RX RZ TOU
FIN
```

Fichier de données pour le module H2, deuxième passage

L'analyse de flambement est lancée par la commande CALcul de FLAmbement, celle-ci correspond à un cas linéaire. Pour pouvoir prendre en compte les influences des imperfections géométriques, cette commande devient alors :

CALcul Flambement GRAnd DEPlacement

Ceci correspond à une première façon de faire un calcul non-linéaire. En effet, en procédant ainsi les déformations géométriques seront prises en compte et il sera aussi possible d'obtenir les efforts correspondants à un cas non-linéaire.

Le fichier suivant du passage au module H6, permet de déterminer les différents coefficients d'éloignement de la charge critique α_{CR}, qui correspond à la résolution du système suivant :

```
/module h6
/
/ Flambement
CAL FLA 1 / 1 mode de flambement
ITE 40
DEP INI 1*0
DEP PRO 101
FIN
```

Fichier de données pour le module H6

Remarque sur les fichiers du module H6 quant au nombre de modes à considérer :
Contrairement au logiciel ROBOT, si le nombre de modes n'est pas correct, une erreur apparaît dans le fichier message comme quoi le nombre de modes doit être diminué. Dans l'exemple de la barre encastrée – articulée, seul *un* mode pouvait être déterminé (car pas de nœuds intermédiaires). Ce qui est équivalent au résultat trouvé par ROBOT.

Le fichier suivant du passage au module H4, permet de sélectionner les résultats:

```
/module h4
/
/ Edition des résultats
     RES CAS
      SEL CAS 2 101
     EFF ELE
     DEP NOE
      SEL NOE G1
     REA NOE
      SEL NOE 1 100
FIN
```

Fichier de données pour le module H4

Dans le cadre d'une procédure d'automatisation, tous les résultats ne sont pas nécessaires, seuls les efforts maxi des barres sont recherchés pour pouvoir dimensionner la structure.

Il est aussi possible d'obtenir les résultats correspondant à un cas particulier (résultat à un chargement ou à un mode de flambement,...) pour un nœud ou une barre.

Les fichiers qui suivent correspondent au module H5, ils permettent de voir sur écran dans un premier temps la structure (et ainsi vérifier qu'elle a bien été définie) et dans un second temps les déformations correspondant au chargement et aux différents modes propres.

```
/module h5
/
/ vérifications : dessins
IMP NOE TOU                / impression des noeuds dans un
                             tableau
IMP ELE TAB NOE CON ORI    / impression des éléments
                             (noeuds, connexions,
                             orientations)
IMP LIA G1                 / impression des liaisons
IMP MAT                    / impression des matériaux

/vérification par deux dessins
/
FIG 1
structure
SOU FIG 1 1 2
NUMEROTATION
DES ELE NUM G1
DES NOE NUM G1
SOU FIG 2
CHARGES ET LIAISONS
DES VAL NOE LEG COO X
DES VAL NOE LEG COO Y
DES VAL NOE LEG COO Z
```

```
DES CHA DET FOR COT G2
DES LIA G1
PRO XZ
FIN
```

Fichier de données pour le module H5, vérification de la géométrie

```
/module h5
/
/ dessins des déformées
FIG 1
  DEFormee FORce
  VECteurs TIRetes
  DES ELE G1
  GEOmetrie DEFormees 2 2
  VECteurs CONtinus
  DES ELE G1
FIG 2
  DEFormee FLAmbement mode 1
  VECteurs TIRetes
  DES ELE G1
  GEOmetrie DEFormees 101 2
  VECteurs CONtinus
  DES ELE G1
FIN
```

Fichier de données pour le module H5, deuxième passage, dessins de la déformée modale

Les fichiers résultats sont de la forme suivante :

```
NUMERO DU MODE * COEFFICIENT DE SECURITE * PRECISION * NUMERO DU CAS
*
        1        *           4.983           *   16.0     *      101
```

Exemple de fichier du module H6,

Dans le résultat de ci-dessus, dans la première colonne, les modes de flambement sont indiqués, ils sont suivis de la valeur du coefficient d'éloignement de la charge critique, «coefficient de sécurité ». La valeur de la précision est indiquée et rapportée à des unités, dans cet exemple la précision est 10^{-4}. La dernière colonne correspond au nom référence qu'Hercule utilise dans le reste des calculs.

```
*                    CAS      2                                       *

        * FORCE

Structure active :  Poutre articulée encastrée


BARRE      1001
NOEUD        1( 0)            NOEUD       2( 0)
No   =  -250.641  TYYo=  0.000       TZZo=  0.000
MXXo=  0.000      MZZo=  0.000       MYYo=  0.000

Ne   =  250.226   TYYe=  0.000       TZZe=  0.000
MXXe=  0.000      MZZe=  0.000       MYYe=  0.000


                                      **      portique      0      noeud**
POUTR  Pas    5  /  Page    3

11    Juin    1999  16:09:15
```

Exemple de fichier de sortie des efforts, du module H4.

Par barre les efforts sont donc fournis pour chacune des extrémités. L'extrémité de départ aura l'indice « $_0$ », alors que l'indice de fin de l'élément est « e ». Théoriquement ce logiciel travaille dans l'espace, et nous avons vu que du fait de la commande « ELIminer COMposante » toutes les informations concernant le plan considéré sont données.

Annex C.

Obtention de l'Expression Mathématique de la Deformation de Green

C. Les déformations de Green dans la non-linéarité

Dans le cadre de notre étude seule une des déformations nous intéresse, il s'agit de la déformation de Green. Bien que la formule générale fut donnée précédemment, ce paragraphe montre les différentes étapes nécessaires pour l'obtention de cette formule. Dans la littérature, deux méthodes différentes permettent la définition de cette déformation.

C.1 Méthode géométrique de détermination de la déformation de Green

1) La formulation de la déformation

La première méthode utilise les coordonnées des points de la barre avant et après déformation.

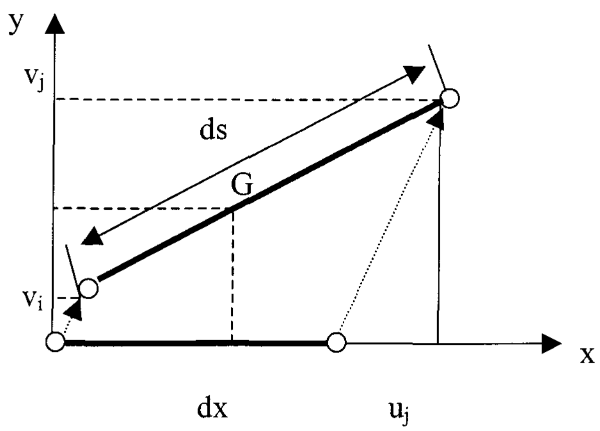


Figure C.1: Détermination de la longueur de la déformation

La barre avant déformation a pour longueur dx, et après déformation ds, sur la figure X.
Cette structure plane, dans le repère général xOy, est composée d'un certain nombre d'éléments finis, éléments qui sont reliés entre eux par des nœuds (fictifs).

Si l'on considère cet élément de longueur non déformé, entre les points i et j, de section A et d'inclinaison Ω par rapport à l'axe Ox.

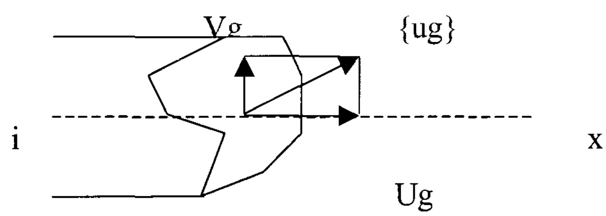


Figure C.2: Section de la bar, vecteurs de l'axe d'inertie

En supposant que l'axe pointillé passe par le centre de gravité G, le vecteur déplacement qui fait passer l'élément de son état initial i vers son état déformé j peut s'exprimer de la façon suivante :

$$\{u_G\} = \begin{bmatrix} U_G \\ V_G \end{bmatrix}$$

U_G et V_G sont les composantes telles qu'indiquées dans la figure Y.
Le repère permet d'exprimer les positions des différents nœuds (des extrémités des barres comme des nœuds fictifs), les déplacements et les charges appliquées à la structure.

Les déformations ε_x de l'élément ij peuvent alors s'exprimer de la façon suivante :

$$\varepsilon_X = \frac{1}{2} \cdot \frac{ds^2 - dx^2}{dx^2} = \frac{(v_j - v_i)^2 + (u_j + dx - u_i)^2 - dx^2}{2 \cdot dx^2} \qquad [A]$$

et si l'on pose les relations suivantes :

$$v_j - v_i = \frac{\partial v}{\partial x} dx$$

$$u_j - u_i = \frac{\partial u}{\partial x} dx$$

l'expression suivante est alors obtenue :

$$\varepsilon_G = \frac{1}{2}\left[\left(\frac{dv}{dx}\right)^2 + \left(\frac{du}{dx} + 1\right)^2 + 1\right]$$

Ceci donne ainsi l'expression de ε_x plus couramment connue sous le nom de déformation de Green.

2) Les déformations de Green

Pour une déformation totale en un point mesuré par rapport à l'axe neutre de l'élément, et en partant de l'origine, la déformation de Green devient :

$$\varepsilon_G = \frac{du}{dx} + \frac{1}{2}\left[\left(\frac{dv}{dx}\right)^2 + \left(\frac{du}{dx}\right)^2\right] \qquad \{1\}$$

Si l'on considère un point quelconque de la barre, la déformation de Green devient alors :

$$\varepsilon_G = \frac{du}{dx} - y \cdot \frac{d^2 v}{dx^2} + \frac{1}{2}\left[\left(\frac{dv}{dx}\right)^2 + \left(\frac{du}{dx}\right)^2\right] \qquad \{2\}$$

3) Exemple pratique

Dans notre étude, en considérant les déformations en un point G de l'axe neutre, et en négligeant les termes du second ordre en u, la déformation devient alors :

$$\varepsilon_{X,G} = \frac{du_G}{dx} + \frac{1}{2} \cdot \left(\frac{dv_G}{dx}\right)^2$$

pour un point M quelconque de la barre, avec les notations suivantes :

$$u_M = u_G - y \frac{dv_G}{dx}$$

$$v_M = v_G$$

L'expression de la déformation du point M devient alors :

$$\varepsilon_{X,G} = \frac{du_G}{dx} + \frac{1}{2} \cdot \left(\frac{dv_G}{dx}\right)^2 - y \cdot \frac{d^2 v_G}{dx^2}$$

C.2 Une méthode alternative pour la détermination de la déformation de Green

Au lieu de définir les déformations à partir du cas des grandes déformations, comme définis dans II.2.1, il est possible de partir directement de la formulation suivante :

$$\varepsilon_G = \frac{l - l_0}{l_0}$$

et si l'on considère la figure II.6 de la partie **1)**, on peut déduire la relation suivante :

$l^2 = (dx + du)^2 + (dv)^2$

avec

$u = u_j - u_i$

$v = v_j - v_i$

donc :

$$\varepsilon(u,v) = \frac{l}{l_0}\sqrt{\left((l_0 + u)^2 + v^2\right)} - 1$$

Par un développement en série, suivant les séries de Taylor, l'expression suivante est obtenue :

$\varepsilon = \varepsilon(0,0) + d\varepsilon/du.u + d\varepsilon/dv.v + ½\ d^2\varepsilon/du^2.u^2 + ½\ d^2\varepsilon/dudv.uv + ½\ d^2\varepsilon/dv^2.v^2$

avec

$d\varepsilon/du = l/l_0$

$d\varepsilon/dv=0$

$d^2\varepsilon/du^2 = 0$

$d^2\varepsilon/dudv = 0$

$d^2\varepsilon/dv^2.v^2 = l/l_0$

Ce qui nous permet d'obtenir l'expression suivante :

$$\varepsilon_{X,G} = \frac{du_G}{dx} + \frac{1}{2}\cdot\left(\frac{dv_G}{dx}\right)^2$$

Il est possible de constater ici que le terme du second ordre en u est manquant. La précision quant à la définition de la déformation à la base en est la cause. Dans le cas de petites déformations cette expression peut tout de même être prise en compte, car c'est justement le terme manquant qui est alors négligé(voir le paragraphe qui va suivre).

Il existe donc certains cas particuliers de la déformation de Green, pour lesquels certains termes sont tellement faibles qu'ils peuvent être négligés.

C.3 Cas particuliers de la déformation de Green

Dans la littérature, trois cas se distinguent et engendrent des simplifications de calculs. Pour illustrer ces différents cas, sera considéré une barre soumise à une déformation d'un champ de déplacement dans le plan (u , v), de la figure de II.2.1, fonction de la coordonnée x seul. Supposons que la déformation soit considérée pour un point placé sur l'axe neutre de déformée :

$$\varepsilon_{G,X} = \frac{du}{dx} + \frac{1}{2}\left[\left(\frac{dv}{dx}\right)^2 + \left(\frac{du}{dx}\right)^2\right]$$

Les différents cas peuvent alors être envisagés et engendrer des simplifications.

1) Cas Linéaire

Les rotations et les déplacements sont très faibles :

$$\frac{du}{dx} \leq 1$$

$$\frac{dv}{dx} \leq 1$$

les termes du second ordre peuvent alors être négligés, et la forme classique suivante est obtenue :

$$\varepsilon_{G,X} \approx \frac{du}{dx}$$

Pour le reste de cette étude, ce cas est appelé : Linéaire du premier ordre.
En France, aux Pays-Bas et en Allemagne ce cas est utilisé pour les « calculs du 1er Ordre ».

2) Cas Non-Linéaire - Grande Rotation – Petite Déformation

La déformation axiale reste petite par rapport aux termes de rotation :

$$\frac{du}{dx} < \frac{dv}{dx}$$

$$\frac{du}{dx} \leq 1$$

d'où la définition suivante :

$$\varepsilon_{G,X} \approx \frac{du}{dx} + \frac{1}{2}\left(\frac{dv}{dx}\right)^2$$

Ce cas est très fréquent dans les études non-linéaires, puisqu'il prend en compte les déplacements importants qui sont engendrés par la rotation du système tout en considérant de petites déformations.
Pour le reste de cette étude, ce cas est appelé : Non-Linéaire du second ordre – Petites déformations.
En France, aux Pays-Bas et en Allemagne ce cas est utilisé pour les « calculs du 2^{e} Ordre, petites déformations ».

3) Cas Non-Linéaire – Grande rotation – Grande Déformation

Les déformations $\frac{du}{dx}$ et les rotations $\frac{dv}{dx}$ sont importantes et ne peuvent plus être négligées. Tous les termes de l'expression de Green doivent alors être pris en compte, d'où la définition suivante, pour un point placé sur l'axe neutre :

$$\varepsilon_{G,X} = \frac{du}{dx} + \frac{1}{2}\left[\left(\frac{dv}{dx}\right)^2 + \left(\frac{du}{dx}\right)^2\right]$$

Pour le reste de cette étude, ce cas est appelé : Non-Linéaire du second ordre – Grandes déformations.
En France, aux Pays-Bas ce cas est utilisé pour les « calculs du 2^{e} Ordre, grandes déformations ».
Alors qu'en Allemagne ce cas est appelé « calculs du 3^{e} Ordre ».

4) Remarque

Le passage du cas 2 (petites déformations) au cas 3 (grandes déformations) n'est pas bien défini dans la littérature. Les Allemands considèrent le 3^e Ordre lorsque les déformations par rapport au dimensionnement de la barre sont importantes. Alors que dans les normes américaines, le 2^{nd} Ordre grandes déformations est pris en compte pour une rotation supérieure à 10.

C.4 Considération graphique de la déformation de Green

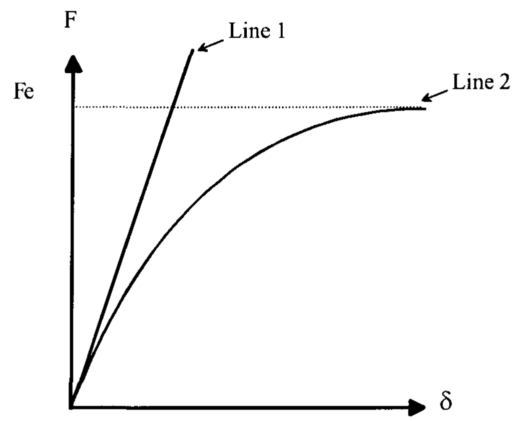


Figure C.3: Répresentation graphique de la déformation Green

Pour donc en revenir au comportement graphique des structures métalliques, et en considérant la déformation de Green, pour un point de l'axe neutre :

$$\varepsilon_G = \frac{du}{dx} + \frac{1}{2}\left[\left(\frac{dv}{dx}\right)^2 + \left(\frac{du}{dx}\right)^2\right]$$

Il est alors possible de dire que la ligne 1 de la figure II.8 correspond à la partie linéaire de la déformation de Green :

u

$$\varepsilon_G = \frac{du}{dx}$$

Cette partie correspondant au premier cas particulier est bien une fonction linéaire.

Alors que la ligne 2 correspond à la déformation complète :

$$\varepsilon_G = \frac{du}{dx} + \frac{1}{2}\left[\left(\frac{dv}{dx}\right)^2 + \left(\frac{du}{dx}\right)^2\right]$$

En plus des termes du premier ordre, des termes du second ordre sont là et cause la non linéarité de la ligne 2.

Annex D.

Outil Mathématique Nécessaire à l'Obtention des Matrices de Rigidité

D. Les matrices de rigidité

Les déformations de Green ont été développées dans le but de comprendre les différents logiciels utilisés pour cette étude. En effet chacun des logiciels est basé sur une procédure matricielle. Pour la détermination de ces matrices la déformation de Green est prise en compte, ainsi que toutes les simplifications qui s'y rattachent.

Dans le cas d'un calcul non-linéaire, il est préférable de faire une étude basée sur les travaux virtuels (Thomas, 1970). Ce qui correspond en fait à une approche par énergie potentielle.

La méthode de l'énergie potentielle est basée sur les déformations (ε). Ces déformations sont fonctions des dérivées des déplacements. Par une analyse d'éléments finis, une expression approximative, pour une structure plane, peut être définie pour les déplacements. Les expressions approximatives sont de formes polynomiales et sont fonctions des coordonnées dans le plan, de l'élément.

Bien que d'autres mesures du comportement, puissent être utilisées, tels les efforts et la solution exacte de l'équation différentielle développée, l'expression ne se limite pas à ce polynôme. Mais, pour l'analyse de la stabilité par éléments finis en pratique, cette forme polynomiale du déplacement supposé est tellement omniprésente dans son rôle qu'elle est tout de même considérée.

D.1 Expression polynomiale de la déformation de Green :

La déformation de Green est définie pour un cas général :

$$\varepsilon_G = \frac{du}{dx} - y \cdot \frac{d^2v}{dx^2} + \frac{1}{2}\left[\left(\frac{dv}{dx}\right)^2 + \left(\frac{du}{dx}\right)^2\right]$$

Le premier terme $\frac{du}{dx}$ à droite de l'inégalité, correspond à la relation linéaire de base en absence d'effets d'instabilités.

Le deuxième terme $y \cdot \frac{d^2v}{dx^2}$ représente la position du point de la barre en fonction de l'axe neutre.

Le troisième terme $\left(\frac{dv}{dx}\right)^2$ représente la déformation axiale due à l'élongation de l'axe neutre dans l'état déformé.

Le quatrième terme $\left(\frac{du}{dx}\right)^2$ peut être considéré comme proportionnel au carré de la déformation axiale et peut être négligé dans les analyses élastiques dans le cas des petites déformations.

D.2 Description du principe de l'énergie potentielle :

L'énergie potentielle est égale à la différence entre l'énergie due aux déformations, U, et l'énergie potentielle due aux forces appliquées, V.
Ep = U - V
D'où :
- la structure est stable pour $E_P > 0$
- la structure est instable pour $E_P < 0$
- le point d'instabilité $E_P = 0$

Dans ce qui va suivre nous allons étudier l'application du principe de l'énergie potentielle dans le cas d'une barre bi-articulée. En considérant la formule de déformation de Green pour un point quelconque de la barre :

1) Energie potentielle interne, U :
L'énergie potentielle interne s'exprime de la façon suivante

$$U = \int_{Vol} \sigma_X \cdot \varepsilon_X d(Vol)$$

avec
ε_X, déformation de Green
σ_X, tenseur des contraintes
Vol, le volume

Sachant que
$\sigma_X = E \,.\, \varepsilon_X$
d(Vol) = dA . dx
Pour cet exemple les termes du second ordre en u seront négligés. L'énergie potentielle interne peut alors s'exprimer comme suit :

$$U = \frac{1}{2}\int_L \int_A \begin{bmatrix} \left(\frac{du}{dx}\right)^2 + y^2 \cdot \left(\frac{d^2v}{dx^2}\right)^2 + \frac{1}{4}\cdot\left(\frac{dv}{dx}\right)^4 - 2y\cdot\frac{du}{dx}\cdot\left(\frac{d^2v}{dx^2}\right) \\ - y\cdot\frac{d^2v}{dx^2}\cdot\left(\frac{dw}{dx}\right)^2 + \frac{du}{dx}\cdot\left(\frac{dw}{dx}\right)^2 \end{bmatrix} \cdot E \cdot dA \cdot dx$$

lors de la résolution de cette intégrale, certaines simplification peuvent avoir lieue :

$$\int_A dA = A$$

$$\int_A y \cdot dA = 0$$

$$\int_A y^2 \cdot dA = I$$

d'où

$$U = \frac{1}{2}\int_L \left[A\cdot\left(\frac{du}{dx}\right)^2 + I\cdot\left(\frac{d^2v}{dx^2}\right)^2 + A\cdot\left(\frac{du}{dx}\right)\cdot\left(\frac{dv}{dx}\right)^2 + \frac{A}{4}\cdot\left(\frac{dv}{dx}\right)^4 \right] \cdot E \cdot dx$$

Comme noté ci-dessus, la représentation par éléments finis, les déplacements des éléments peuvent être approchés. Les expressions des déplacements (en u et v) peuvent être écrites de la façon suivante :

$$u=\sum_{i=1}^{n} N_i^u \cdot u_i = \lfloor N^u \rfloor \{u\}$$

$$v=\sum_{i=1}^{n} N_i^v \cdot v_i + \sum_{i=1}^{n} N_i^\theta \cdot \theta_i = \lfloor N^v N^\theta \rfloor \begin{Bmatrix} v \\ \theta \end{Bmatrix}$$

avec

N_i^v, N_i^θ et N_i^u fonctions des coordonnées x et sont appelées « fonctions de forme ».

[N^w N^θ] et [N^u] sont les matrices en ligne contenant ces fonctions.

u_i et v_i représentent les déplacements au niveau du nœud i.

θ_i représente le déplacement angulaire au niveau du nœud i

{w}, {u} et {θ} représente le listing en colonne des déplacements des points d'assemblage.

Il est intéressant de noter qu'apparemment le terme linéaire de la déformation de Green n'est fonction que des forces axiales. Alors que les autres termes dépendent des déplacements normaux et des rotations.

Après substitution des expressions de u et v, et après intégration, le résultat suivant est obtenu :

$$U=\frac{\lfloor\Delta\rfloor}{2}\cdot[k_0]\cdot\{\Delta\}+\frac{\lfloor\Delta\rfloor}{6}\cdot[k_1]\cdot\{\Delta\}+\frac{\lfloor\Delta\rfloor}{12}\cdot[k_2]\cdot\{\Delta$$

$$U=\frac{\lfloor\Delta\rfloor}{2}\cdot[k_S]\cdot\{\Delta\}$$

Avec

$[k_S] = [k_0] + [k_1] + [k_2]$

[Δ] = [[u][v θ]]

$[k_S]$: représente la « matrice de rigidité sécante », d'un cas non linéaire.

$[k_0]$: représente la matrice de rigidité linéaire, d'un état stable.

$[k_1]$: représente la matrice des « efforts initiaux », aussi appelé « géométrique ». Cette matrice tient compte de la fonction linéaire des forces N ainsi que des rotations θ.

$[k_2]$: représente la matrice des « déplacements initiaux ». Cette matrice tient compte de la fonction quadratique des rotations θ.

Il est possible de remarquer que si l'on se place dans le cas linéaire, la matrice de rigidité sécante $[k_S]$, ne tiendra compte que des matrices $[k_0]$ et $[k_1]$. Car alors les déplacements ne sont pas pris en compte dans l'analyse.

La matrice k_S définie ci-dessus n'est le résultat que d'un cas particulier d'une barre. Pour une structure complète, l'énergie potentielle totale est égale à la somme de toutes les déformations des éléments de cette structure :

$$U=\sum U=\frac{\lfloor\Delta\rfloor}{2}\cdot[K_S]\cdot\{\Delta\}$$

C'est donc [Δ] qui détermine les déplacements de tous les nœuds, et $[K_S]$ représente la matrice de rigidité totale.

Dans le cas des études Non-Linéaire, cette matrice $[K_S]$ est appelé « matrice de rigidité sécante ».

2) Energie potentielle externe, V :
Si l'on considère des forces concentrées sur les nœuds, l'énergie potentielle causée par les forces appliquées peut s'écrire :
$V = [\Delta]\ \{P\}$
Avec
$\{P\}$ la liste des forces au niveau des nœuds d'assemblages.
$[\Delta]$ = matrice de déplacement des nœuds

3) Energie potentielle totale, E_p :
D'après les solutions issues de 1) et 2), l'énergie potentielle totale de la forme suivante :
$E_P = U - V$
Devient :
$$E_P = \frac{\lfloor \Delta \rfloor}{2} \cdot [K_S] \cdot \{\Delta\} - \lfloor \Delta \rfloor \cdot \{P\}$$
Ce qui nous intéresse alors est de trouver le point d'instabilité, le système $E_p = 0$ doit être résolu. D'où :
$$\left[[K_0] + \frac{1}{2} \cdot [K_1] + \frac{1}{3} \cdot [K_2] \right] \cdot \{\Delta\} - \{P\} = 0$$
ou
$$[[K]] \cdot \{\Delta\} - \{P\} = 0$$

La différence entre la matrice sécante $[K_S]$ et la matrice précédemment définie $[K]$ se situe dans les coefficients de multiplication des termes $[K_1]$ et $[K_2]$; les résultats du processus de différentiation affectent les matrices, car elles sont fonctions des déplacements.

Dans la plupart des problèmes non-linéaire, ce sont surtout les différentiations qui sont considérées, l'expression devient alors :
$$[K_T] \cdot \{\delta\Delta\} - \{\delta P\} = 0$$
avec
$[K_T]$, matrice de rigidité tangente, telle que
$[K_T] = [K_0] + [K_\sigma] + [K_D]$

Cette matrice K_T se décompose en trois matrices distinctes :
K_0, appelé matrice de rigidité initiale (matrice de petits déplacements)(notée K_0).
K_σ, appelé matrice de contrainte initiale.
K_D, appelé matrice de grand déplacement.

D.3 Exemple de matrices

La matrice de rigidité tangente peut s'écrire de la façon suivante :

$$[K_T]=\begin{bmatrix} \frac{EA}{L} & 0 & 0 & -\frac{EA}{L} & 0 & 0 \\ & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ & & \frac{4EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{2EI}{L^2} \\ & & & \frac{EA}{L} & 0 & 0 \\ & \mathit{Symétrique} & & & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ & & & & & \frac{4EI}{L} \end{bmatrix}$$

[MUZE,79], exemple de trois différentes matrices dans le cas d'une barre:

$$[K_0]=\begin{bmatrix} 0 & 0 & 0 & -\frac{EA}{L} & 0 & 0 \\ & \frac{12EI}{L^3} & \frac{6EI}{L^2} & 0 & -\frac{12EI}{L^3} & \frac{6EI}{L^2} \\ & & \frac{4EI}{L} & 0 & -\frac{6EI}{L^2} & \frac{2EI}{L} \\ & & & \frac{EA}{L} & 0 & 0 \\ & \mathit{Symétrique} & & & -\frac{12EI}{L^3} & -\frac{6EI}{L^2} \\ & & & & & \frac{4EI}{L} \end{bmatrix}$$

$$[K_\sigma]=\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ & -\frac{3}{5L}(N_i+N_j) & -\frac{N_j}{10} & 0 & \frac{3}{5L}(N_i+N_j) & -\frac{N_i}{10} \\ & & -\frac{3N_i+N_j}{30}L & 0 & \frac{N_j}{10} & \frac{N_i+N_j}{60}L \\ & & & 0 & 0 & 0 \\ & \mathit{Symétrique} & & & -\frac{3}{5L}(N_i+N_j) & \frac{N_i}{10} \\ & & & & & -\frac{N_i+3N_j}{30}L \end{bmatrix}$$

$$[K_D]=\begin{bmatrix} 0 & \alpha & \beta & 0 & -\alpha & \gamma \\ \alpha & \frac{\delta}{7} & \frac{\varepsilon}{7} & -\alpha & -\frac{\delta}{7} & \frac{\upsilon}{7} \\ \beta & \frac{\varepsilon}{7} & \frac{\Phi}{7} & -\beta & -\frac{\varepsilon}{7} & \frac{\lambda}{7} \\ 0 & -\alpha & -\beta & 0 & \alpha & -\gamma \\ -\alpha & -\frac{\delta}{7} & -\frac{\varepsilon}{7} & \alpha & \frac{\delta}{7} & -\frac{\upsilon}{7} \\ \gamma & \frac{\upsilon}{7} & \frac{\lambda}{7} & -\gamma & -\frac{\upsilon}{7} & \frac{\mu}{7} \end{bmatrix}$$

Avec :

$$\alpha = \frac{6}{L}(v_j - v_i) - \frac{\beta_i - \beta_j}{2L}$$
$$\beta = \frac{(v_j - v_i)}{2L} - \frac{4\beta_i - \beta_j}{6}$$
$$\gamma = \frac{(v_j - v_i)}{2L} - \frac{\beta_i - 4\beta_j}{6}$$
$$\delta = \frac{72}{L^3}(v_i - v_j)^2 + \frac{18}{L}(\beta_i - \beta_j)(v_i - v_j) + \frac{3}{L}(\beta_i^{\ 2} + \beta_j^{\ 2})$$
$$\varepsilon = \frac{9}{L^2}(v_i - v_j)^2 + \frac{6}{L}\beta_i(v_i - v_j) + \frac{1}{4}(\beta_j^{\ 2} - \beta_i^{\ 2} + 2\beta_i\beta_j)$$
$$\Phi = \frac{3}{L}(v_i - v_j)^2 - \frac{1}{2}(\beta_i - \beta_j)(v_i - v_j) + \frac{L}{6}(12\beta_i^{\ 2} + \beta_j^{\ 2} - 3\beta_i\beta_j)$$
$$\upsilon = \frac{9}{L^2}(v_i - v_j)^2 + \frac{6}{L}\beta_j(v_i - v_j) + \frac{1}{4}(\beta_i^{\ 2} - \beta_j^{\ 2} + 2\beta_i\beta_j)$$
$$\lambda = \frac{1}{2}(\beta_i + \beta_j)(v_i - v_j) + \frac{L}{12}(-3\beta_i^{\ 2} - 3\beta_j^{\ 2} + 4\beta_i\beta_j)$$
$$\mu = \frac{3}{L}(v_i - v_j)^2 - \frac{1}{2}(\beta_i - \beta_j)(v_i - v_j) + \frac{L}{6}(\beta_i^{\ 2} + 12\beta_j^{\ 2} - 3\beta_i\beta_j)$$

Sans trop entrer dans les détails de ces trois matrices, il est néanmoins possible de confirmer les faits suivant :

la matrice K_0 n'est dépendante que des termes de raideur de la barre
la matrice K_σ n'est dépendante que de l'action N appliquée à la barre
la matrice K_D ne prend en compte que des termes du second ordre

D.4 Autre décomposition possible de K_T

Dans la littérature les matrices qui composent la matrice de rigidité tangente semblent différentes. En effet dans [PEP,95], on constate que la matrice de rigidité tangente est exprimée de la façon suivante :
$R = R_0(N^*) + R_1(N^*) + R_1(\Phi^*) + R_2(\Phi^*)$

Avec :
- $R_0(N^*)$: matrice principale dont les termes de rigidité font intervenir les fonctions dites de « stabilité » qui introduisent la perte de rigidité flexionnelle de la barre due à la compression seule ou le gain de rigidité flexionnelle dû à la traction.
- $R_1(N^*)$: matrice dite « géométrique », fonction linéaire de N*, introduisant des efforts stabilisants ou déstabilisants (selon le signe de N*) aux extrémités de la barre dus à une variation de la rotation globale de celle-ci.
- $R_1(\Phi^*)$: matrice fonction linéaire de Φ* actualisant la géométrie de la structure (déplacement simulé du repère local de la barre).
- $R_2(\Phi^*)$: matrice fonction quadratique de Φ* actualisant également la géométrie de la structure (déplacement simulé du repère local de la barre).

Les effets du second ordre sont introduit par l'effort normal de la barre
$N^* = N + N_0$ (N_0 : effort normal initial)

Et sa « rotation globale totale » :
$\Phi^* = \Phi + \Phi_0$ (Φ_0 : rotation globale initiale)
La rotation globale étant par définition la rotation de la corde liant les extrémités de la barre. (Le principe étant basé sur les faibles rotations).
Il est alors possible de se demander s'il y a une différence entre les deux matrices K_T et R ?
Non, les matrices K_T et R sont identiques. Les termes de ces matrices sont les mêmes.

La différence, entre KT et R, tient surtout dans la définition des termes qui composent les matrices de rigidité initiale$[K_0]$, d'effort initial $[K_\sigma]$ et de déplacement$[K_D]$.

En effet, pour définir K_T, la position des nœuds prend une part importante. Tous les termes sont donc fonction de (u_i , v_i) et (u_j , v_j).
Dans le cas de la matrice R, ce sont les efforts et les rotations qui sont indiqués dans les matrices.
Il existe donc une relation entre les matrices K_T et R. notamment :

$$N = \frac{EA}{L}(u_j - u_i)$$

$$\Phi = \frac{dv}{dx}$$

et l'on peut donc dire que

$[R_0]=[K_0]$

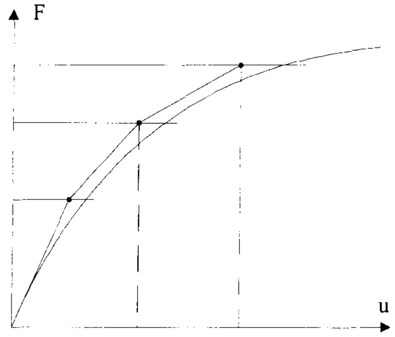
$[R_1]=[K_\sigma]$

$[R_2]=[K_D]$

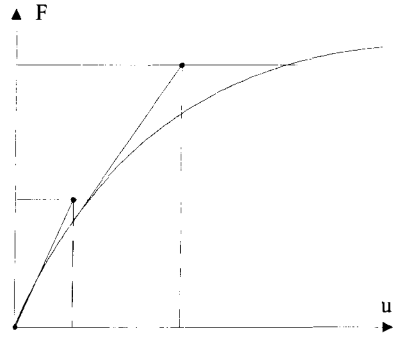
Annex E.

Différents Outils Permettant d'Approcher le Comportement Non-linéaire

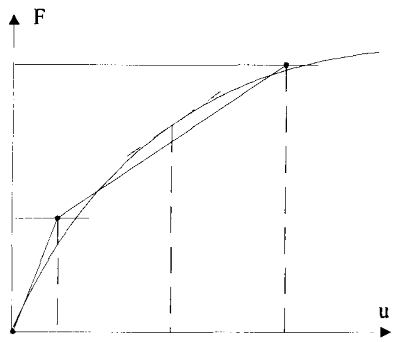
E. Tools to solve non-linear equations



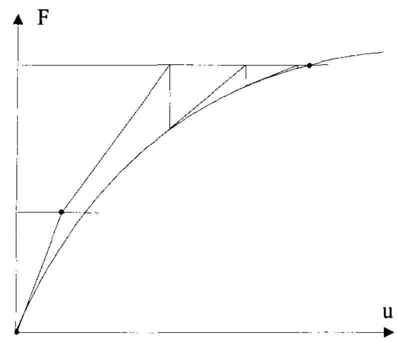
1



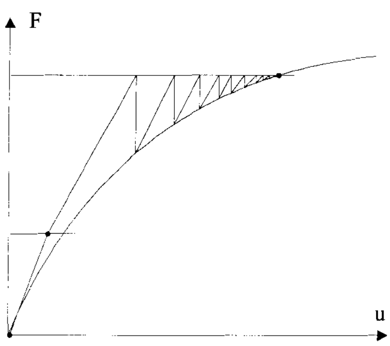
2



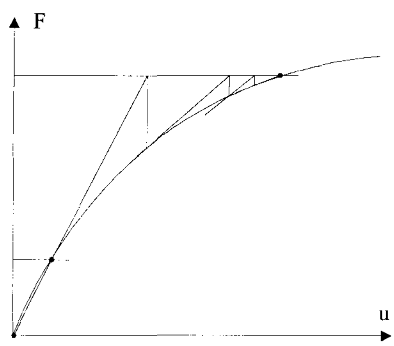
3



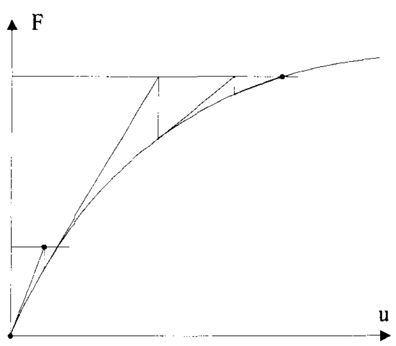
4



5



6



7

→

<u>Tools to solve non-linear equations :</u>

1. Step by step method
2. Step with residual load
3. Mid-point method
4. Newton-Raphson method
5. Modified Newton-Raphson method
6. Secant iteration method
7. Newton-Raphson method with residual load