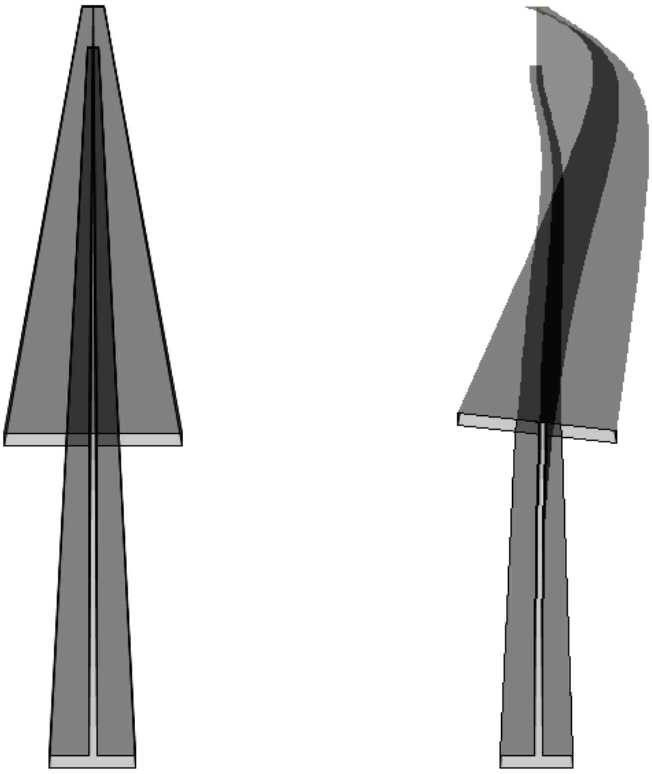


# Lateral-Torsional Buckling of Steel Beams with Open Cross Section

## Elastic Critical Moment Study and Software Application

*Master of Science Thesis in the Master's Programme Structural Engineering and Building Technology*


HERMANN ÞÓR HAUKSSON
JÓN BJÖRN VILHJÁLMSSON

Department of Civil and Environmental Engineering
*Division of Structural Engineering*
*Steel and Timber Structures*
CHALMERS UNIVERSITY OF TECHNOLOGY



Göteborg, Sweden 2014
Master's Thesis 2014:28

MASTER'S THESIS 2014:28

# Lateral-Torsional Buckling of Steel Beams with Open Cross Section

Elastic Critical Moment Study and Software Application

*Master of Science Thesis in the Master's Programme Structural Engineering and Building Technology*

HERMANN ÞÓR HAUKSSON

JÓN BJÖRN VILHJÁLMSSON

Department of Civil and Environmental Engineering
*Division of Structural Engineering*
*Steel and Timber Structures*
CHALMERS UNIVERSITY OF TECHNOLOGY

Göteborg, Sweden 2014

Lateral-Torsional Buckling of Steel Beams with Open Cross Section
Elastic Critical Moment Study and Software Application

*Master of Science Thesis in the Master's Programme Structural Engineering and Building Technology*

HERMANN ÞÓR HAUKSSON
JÓN BJÖRN VILHJÁLMSSON



Examensarbete / Institutionen för bygg- och miljöteknik,
Chalmers tekniska högskola 2014:28

Department of Civil and Environmental Engineering
Division of Structural Engineering
Steel and Timber Structures
Chalmers University of Technology
SE-412 96 Göteborg
Sweden
Telephone: + 46 (0)31-772 1000

Cover:
Lateral-torsional buckling of a monosymmetric I-beam, as illustrated by the commercial structural engineering software SAP2000

Chalmers Reproservice
Göteborg, Sweden 2014

Lateral-Torsional Buckling of Steel Beams with Open Cross Section
Elastic Critical Moment Study and Software Application

*Master of Science Thesis in the Master's Programme Structural Engineering and Building Technology*

HERMANN ÞÓR HAUKSSON
JÓN BJÖRN VILHJÁLMSSON
Department of Civil and Environmental Engineering
Division of Structural Engineering
Steel and Timber Structures
Chalmers University of Technology

ABSTRACT

The elastic critical moment $M_{cr}$ is an important design parameter when it comes to lateral-torsional buckling of steel beams. Lateral-torsional buckling is influenced by number of factors and thus structural engineering software often produce different results, based on different assumptions. A monosymmetric I-beam was studied considering different load heights and different degree of end-restraint for rotation about the minor axis and warping. Beams with a channel cross section were studied considering different beam lengths, different types of load and different eccentricities of the loads, including centric loading. $M_{cr}$ was calculated using various software and compared to values assessed by an analytical expression called the 3-factor formula. To obtain the elastic critical moment for eccentrically loaded channel beams, a number of methods were used.

The difference between the software tools studied was based on different methods and assumptions they use to calculate $M_{cr}$. For example not all of them take the monosymmetry of the cross section into account while others only count on the beam bending moment diagram. Some use the finite element method while others use analytical expressions. The methods implemented in the current thesis were not successful in obtaining the elastic critical moment for eccentrically loaded channel beams. No clear instability behaviour was detected unless the beams were loaded in the vertical plane through its shear centre.

The software tools used in this thesis are ADINA, Colbeam, LTBeam, SAP2000 and STAAD.Pro.

Key words: Lateral-torsional buckling, elastic critical moment, 3-factor formula, warping, torsion, fork supports, monosymmetric I-section, channel section, ADINA, Colbeam, LTBeam, SAP2000, STAAD.Pro

# Contents

# Preface

This master's thesis project was carried out in 2014 between January and June at Reinertsen Sweden AB in cooperation with the Department of Civil and Environmental Engineering, Division of Structural Engineering, Steel and Timber Structures at Chalmers University of Technology in Göteborg, Sweden. Professor Robert Kliger at the Division of Structural Engineering was the examiner of the thesis.

Supervisors of the thesis were Martin Gustafsson, Structural Engineer at Reinertsen Sweden AB and Professor emeritus Bo Edlund at the Division of Structural Engineering. To them we would like to give thanks for their continuous support and guidance during the thesis project.

We also thank our opponents Bengt Lundquist and Guðni Ellert Edvardsson for the discussions they had with us about the subject and providing thoughtful comments, as well as proofreading the thesis.

At last we thank the staff at Reinertsen Sweden AB for providing us a comfortable working environment and support, especially concerning the various commercial software tools employed in this thesis.

Göteborg June 2014

Hermann Þór Hauksson & Jón Björn Vilhjálmsson

# Notations

### Abbreviations

| | |
|---|---|
| CISC | Canadian Institute of Steel Construction |
| CTICM | Centre Technique Industriel de la Construction Métallique |
| DOFs | Degrees Of Freedom |
| ECCS | European Convention for Constructional Steelwork |
| FEM | Finite Element Method |
| GC | Centre of Gravity |
| LDC | Load-Displacement-Control |
| LT-buckling | Lateral-torsional buckling |
| NCCI | Non Contradictory Complementary Information |
| OP | Origin of coordinate system |
| PLA | Point of Load Application |
| SC | Shear Centre |
| TC | Torsion Centre |

### Roman upper case letters

| | |
|---|---|
| $A$ | Area of the cross section |
| $A_1$ | Value to determine the $C_1$ factor using the closed form expression to calculate $M_{cr}$ |
| $A_{15}$ | Wagner effect constant |
| $A_2$ | Value to determine the $C_1$ factor using the closed form expression to calculate $M_{cr}$ |
| $A_{25}$ | Wagner effect constant |
| $A_{35}$ | Wagner effect constant |
| $A_{45}$ | Wagner effect constant |
| $A_{55}$ | Wagner effect constant |
| $A_f$ | Area of the flange in compression |
| $A_{w,c}$ | Area of the part of the web which is in compression |
| $B$ | Bi-moment |
| $C_1$ | Factor depending on the moment diagram and the end restraints |
| $C_2$ | Factor depending on the moment diagram and the end restraints, related to the vertical position of loading |
| $C_3$ | Factor depending on the moment diagram and the end restraints, related to the monosymmetry of the beam |

| | |
|---|---|
| $D_1$ | Value to determine the Wagner's coefficient $\beta_z$ |
| $D_2$ | Value to determine the Wagner's coefficient $\beta_z$ |
| $E$ | Young's modulus of elasticity |
| $EI_w$ | Warping stiffness |
| $F_{i-1}^{t+\Delta t}$ | Nodal force vector at time $t+\Delta t$ and end of iteration number $i-1$ |
| $G$ | Shear modulus |
| $GI_t$ | Torsional stiffness |
| $H$ | Total potential energy |
| $\Delta H$ | Change in the total potential energy |
| $H_{deflected}$ | Total potential energy in the deflected equilibrium position |
| $H_e$ | External potential energy |
| $H_i$ | Internal potential energy |
| $H_{straight}$ | Total potential energy in the straight equilibrium position immediately before buckling |
| $I_f$ | Second moment of area of the flange in compression about the minor axis |
| $I_{f1}$ | Second moment of area for the top flange about the minor axis |
| $I_{f2}$ | Second moment of area for the bottom flange about the minor axis |
| $I_t$ | Saint-Venant torsional constant |
| $I_w$ | Warping constant |
| $I_y$ | Second moment of area about the major axis |
| $I_{yz}$ | Product of second moment of area |
| $I_z$ | Second moment of area about the minor axis |
| $K_G$ | Geometrical matrix |
| $K_L$ | Linear matrix |
| $K^{t_0}$ | Stiffness matrix of the structural system at time $t_0$ |
| $K^{t_1}$ | Stiffness matrix of the structural system at time $t_1$ |
| $K_{NL}^{t_1}$ | Geometrically nonlinear part of the stiffness matrix $K^{t_1}$ |
| $K_{i-1}^{t+\Delta t}$ | Tangent stiffness matrix at time $t+\Delta t$ and end of iteration number $i-1$ |
| $L$ | Length of the beam between points which have lateral restraints |
| $L_{cr}$ | Buckling length |
| $\overline{M}$ | Distribution of the bending moment along the beam |
| $M_1$ | Moment value acting on a beam at the left end seen from the user |
| $M_2$ | Moment value acting on a beam at the point which is $0.25L$ from the left end seen from the user |
| $M_3$ | Moment value acting on a beam at the centre of the span |

| | |
|---|---|
| $M_4$ | Moment value acting on a beam at the point which is $0.25L$ from the right end seen from the user |
| $M_5$ | Moment value acting on a beam at the right end seen from the user |
| $M_{Ed}$ | Design bending moment |
| $M_{b,Rd}$ | Design buckling resistance moment |
| $M_{c,Rd}$ | Design resistance for bending about one principal axis of a cross section |
| $M_{cr}$ | Elastic critical moment for lateral-torsional buckling |
| $M_{cr,c}$ | Elastic critical moment for eccentrically loaded channel beams taking the design method proposed by Snijder et al. (2008), described in Section 2.5.2, into account |
| $M_{fl}$ | Moment in the flanges when calculating the bi-moment for a beam with an I-section |
| $M_{max}$ | Maximum moment acting on a beam |
| $M_{pl}$ | Plastic moment capacity of the cross section |
| $M_{y,Ed}$ | Design bending moment about the y-axis |
| $M_{y,Rk}$ | Characteristic bending resistance about the y-axis |
| $R$ | Constant reference load vector |
| $R_p$ | Load vector from previous solution run |
| $T$ | Torsion |
| $U_i^{t+\Delta t}$ | Displacement vector at time $t + \Delta t$ and end of iteration number $i$ |
| $\Delta U_i$ | Incremental displacement vector in iteration number $i$ |
| $W_y$ | Appropriate section modulus corresponding to the compression flange, depending on the cross section class |
| $X$ | Eigenvector |

**Roman lower case letters**

| | |
|---|---|
| $a$ | Value to determine the warping constant for channel sections |
| $b_f$ | Width of the beam |
| $b_{f1}$ | Width of the top flange |
| $b_{f2}$ | Width of the bottom flange |
| $b_s$ | Width of the flanges to the centre of the web |
| $c$ | Torsional constant multiplier |
| $e$ | Distance from the shear centre to the centre of the web |
| $e_1$ | Distance between the top flange and the shear centre of the section |
| $e_2$ | Distance between the bottom flange and the shear centre of the section |

| | |
|---|---|
| $f_y$ | Yield strength |
| $h$ | Height of the beam |
| $h_i$ | Length of cross section part $i$ |
| $h_s$ | Distance between the shear centre of the top flange and the shear centre of the bottom flange of a beam |
| $h_w$ | Height of the web |
| $i_{f,z}$ | Radius of gyration of the equivalent compression flange composed of the compression flange plus $1/3$ of the compressed part of the web area, about the minor axis of the section |
| $k$ | Buckling length factor dependent on $k_1$ and $k_2$ |
| $k_1$ | Buckling length factor on the left end seen from the user, representing $k_w$ and $k_z$ |
| $k_2$ | Buckling length factor on the right end seen from the user, representing $k_w$ and $k_z$ |
| $k_c$ | Slenderness correction factor for the moment distribution between restraints |
| $k_{f\ell}$ | Modification factor accounting for the conservatism of the equivalent compression flange method |
| $k_w$ | Effective length factor which refers to end warping |
| $k_y$ | Effective length factor which refers to end rotation about the major axis |
| $k_z$ | Effective length factor which refers to end rotation in plan (about the minor axis) |
| $n$ | Number of cross section parts for a cross section built up by rectangular parts |
| $m_x$ | Distributed torsional moment along the beam |
| $t_f$ | Thickness of the beam flanges |
| $t_{f1}$ | Thickness of the top flange |
| $t_{f2}$ | Thickness of the bottom flange |
| $t_i$ | Thickness of cross section part $i$ |
| $t_w$ | Thickness of the web |
| $u_{\mathrm{x}}$ | Translational degree of freedom in the x-direction |
| $u_{\mathrm{y}}$ | Translational degree of freedom in the y-direction |
| $u_{\mathrm{z}}$ | Translational degree of freedom in the z-direction |
| $y_a$ | Coordinate of the point of load application with respect to the centre of gravity in the y-direction |
| $y_g$ | Coordinate of the point of load application w.r.t. the shear centre in the y-direction |

| | |
|---|---|
| $y_s$ | Coordinate of the shear centre w.r.t. the centre of gravity in the y-direction |
| $z_G$ | Distance from the centre of gravity to the bottom of the cross section |
| $z_a$ | Coordinate of the point of load application with respect to the centre of gravity in the z-direction |
| $z_g$ | Coordinate of the point of load application w.r.t. the shear centre in the z-direction |
| $z_j$ | Monosymmetry parameter |
| $z_s$ | Coordinate of the shear centre w.r.t. the centre of gravity in the z-direction |

**Greek upper case letters**

| | |
|---|---|
| $\Phi_{LT}$ | Value to determine the reduction factor $\chi_{LT}$ |

**Greek lower case letters**

| | |
|---|---|
| $\alpha_1$ | Value to determine the $C_1$ factor using the closed form expression to calculate $M_{cr}$ |
| $\alpha_2$ | Value to determine the $C_1$ factor using the closed form expression to calculate $M_{cr}$ |
| $\alpha_3$ | Value to determine the $C_1$ factor using the closed form expression to calculate $M_{cr}$ |
| $\alpha_4$ | Value to determine the $C_1$ factor using the closed form expression to calculate $M_{cr}$ |
| $\alpha_5$ | Value to determine the $C_1$ factor using the closed form expression to calculate $M_{cr}$ |
| $\alpha_{LT}$ | Imperfection factor, lateral-torsional buckling curves |
| $\alpha_{cr,op}$ | Minimum amplifier for the in plane design loads to reach the elastic critical resistance with regard to flexural buckling due to minor axis bending and lateral-torsional buckling |
| $\alpha_{ult,k}$ | Minimum load amplifier of the design loads to reach the characteristic resistance of the most critical cross section |
| $\beta$ | Correction factor for the lateral-torsional buckling curves for rolled sections |
| $\beta_y$ | Wagner's coefficient describing the degree of symmetry about the z-axis |
| $\beta_z$ | Wagner's coefficient describing the degree of symmetry about the y-axis |
| $\gamma_i$ | Eigenvalue for mode $i$ |
| $\gamma_{M1}$ | Partial factor for the resistance of the member to instability assessed by member checks |
| $\bar{\lambda}_1$ | Slenderness value to determine the relative slenderness |
| $\lambda_i$ | Critical buckling load factor for mode $i$ |

| | |
|---|---|
| $\lambda_{i-1}^{t+\Delta t}$ | Load scaling factor applied on the constant reference load vector $R$ at time $t + \Delta t$ and end of iteration number $i - 1$ |
| $\Delta\lambda_i$ | Increment in the load scaling factor in iteration number $i$ |
| $\bar{\lambda}_{LT}$ | Non-dimensional slenderness for lateral-torsional buckling |
| $\bar{\lambda}_{LT,0}$ | Plateau length of the lateral-torsional buckling curves for rolled sections |
| $\bar{\lambda}_{MT}$ | Non-dimensional slenderness for buckling with influence from torsion |
| $\bar{\lambda}_{M}$ | Non-dimensional slenderness for lateral-torsional buckling of channel beams |
| $\bar{\lambda}_{T}$ | Additional non-dimensional slenderness for lateral-torsional buckling of channel beams due to torsion |
| $\bar{\lambda}_{c0}$ | Slenderness parameter |
| $\bar{\lambda}_{f}$ | Equivalent compression flange slenderness |
| $\bar{\lambda}_{op}$ | Global non-dimensional slenderness of a structural component for out-of-plane buckling |
| $\mu_{cr}$ | Critical value of the multiplier $\mu$ |
| $\nu$ | Poisson's ratio |
| $\sigma_{xx}$ | Axial stress in the x-direction |
| $\varphi$ | Angle of twist |
| $\varphi_x$ | Rotational degree of freedom about the x-axis |
| $\varphi_y$ | Rotational degree of freedom about the y-axis |
| $\varphi_z$ | Rotational degree of freedom about the z-axis |
| $\phi_i$ | Buckling mode shape for mode $i$ |
| $\chi$ | Reduction factor of the equivalent compression flange determined with $\bar{\lambda}_f$ |
| $\chi_{LT}$ | Reduction factor for lateral-torsional buckling |
| $\chi_{op}$ | Reduction factor for the non-dimensional slenderness $\bar{\lambda}_{op}$ |
| $\psi$ | Warping function |
| $\psi_f$ | Differential ratio of the second moments of area about the minor axis for the beam flanges |
| $\omega$ | Warping degree of freedom |

# 1 Introduction

One of the tasks engineers face, when designing beams with an open cross section is lateral-torsional buckling, often referred to as LT-buckling. LT-buckling can occur in major axis bending of a beam, where the stiffness about the minor axis is relatively small in comparison to the stiffness about the major axis. Before the steel yields, the compression flange buckles in the transversal direction to the load, pulling the beam sideways, while the flange in tension tends to hold the beam in place. This is called lateral-torsional buckling and the steel beam is no longer suitable for its original purpose.

When designing with respect to LT-buckling according to the design code EN 1993-1-1:2005, hereinafter referred to as Eurocode 3, one parameter to be noticed is the relative slenderness of the beam. It is determined from two basic parameters; the plastic moment capacity of the cross section, $M_{pl}$, and the elastic critical moment, $M_{cr}$. The lower $M_{cr}$ is, the higher the relative slenderness will be. A higher relative slenderness implies a lower reduction factor, and the design moment capacity of the beam reduces. However, there is nothing stated about how to determine $M_{cr}$ in Eurocode 3. The factors influencing $M_{cr}$ according to analytical expressions are:

- The stiffness about the minor axis
- The torsional stiffness
- The warping stiffness
- The length of the beam
- The boundary conditions
- The type of the load
- The vertical position of the loading
- Material parameters
- Degree of symmetry about the major axis

A considerable number of present-day commercial structural engineering software take lateral-torsional buckling into account, when evaluating the capacity of steel beams. This means that it is possible for a designer to select a beam without putting an effort into understanding the LT-buckling mechanism. Some software offer the possibility to design beams with a channel section with regard to lateral-torsional buckling, but the design rules are only valid if the channel beams are centrically loaded (Snijder et al. 2008).

Engineers at Reinertsen Sweden AB have noticed different results for $M_{cr}$ depending on which software is used, even for the simplest cases. When the difference is significant, it makes the engineer raise doubts about the reliability of the results. For that reason, a good understanding of the problem at hand, as well as of the methods and assumptions that the available software use to obtain their results is vital.

As a continuation of a previous work, where beams with a doubly symmetric I-section were studied (Ahnlén & Westlund 2013), it is of interest to study beams with a monosymmetric I-section. There exists an analytical expression, called the *3-factor formula*, which is applicable for estimating $M_{cr}$ for beams with an I-section. The formula is based on various parameters such as sectional stiffness parameters, material parameters and effective buckling length parameters. In order to take different loading and support conditions into account, three multiplication factors are applied and referred to as the C-factors.

In steel structures, beams with channel sections are often used and loaded on the web or the flanges. The shear centre (see Section 2.1.5) of channel sections is, on the other hand, located outside the cross section, which means that beams with such a section are often eccentrically loaded in practice. However, Eurocode 3 does not treat lateral-torsional buckling of eccentrically loaded beams (Snijder et al. 2008).

It is of special interest to study the behaviour of beams with a channel section, either subjected to centric or eccentric loading, and see in what way different software treat such beams. Evaluation of $M_{cr}$ for channel beams is of concern and preferably by using an approximate formula or a task-specific software, so it would be suitable to employ in design.

## 1.1 Aim of the thesis

The aim is to investigate how the elastic critical moment, $M_{cr}$, is calculated for beams with a channel section or a monosymmetric I-section, using five chosen commercial software, and investigate how $M_{cr}$ can be found for eccentrically loaded channel beams.

## 1.2 Objectives

To be able to meet the aim of the thesis the following objectives are presented, with tasks that need to be performed:

- Understand the instability behaviour of steel members and the theory behind lateral-torsional buckling.
- Find and study analytical expressions for $M_{cr}$.
- Explore the methods and assumptions used in each of the chosen commercial software.
- Create appropriate FEM models of beams with a channel section and a monosymmetric I-section in ADINA.
- Obtain $M_{cr}$ by performing suitable analyses of the models under different conditions and perform comparison between different methods, and to reference values where possible.

## 1.3 Method

A literature study will be performed in order to understand the mechanism behind lateral-torsional buckling and the evaluation of $M_{cr}$. Analytical expressions for $M_{cr}$ of beams and their inputs will be studied. Those expressions are investigated for I-beams with doubly symmetric cross section before moving on to monosymmetric I-beams and finally channel beams.

The five commercial structural engineering software ADINA, Colbeam, LTBeam, SAP2000 and STAAD.Pro will be used for modelling in order to find $M_{cr}$. First a doubly symmetric I-beam will be modelled in the different software in order to confirm the modelling method is appropriate. Then a monosymmetric I-beam will be modelled using different lateral restraints with a point load applied at various heights and the resulting $M_{cr}$ compared.

A channel beam will be modelled, using the various software that provide that possibility, and its $M_{cr}$ calculated for different load cases and beam lengths. In ADINA the channel beam will be modelled using shell elements, so special care must be taken when it comes to modelling support conditions and load application for instance. The obtained results for $M_{cr}$ will be compared between programs and to results from analytical expressions and reference values if possible.

The channel beam will also be subjected to eccentric loading and an attempt will be made to obtain $M_{cr}$ using various methods. For the shell element model in ADINA, a linearized buckling analysis and a collapse analysis with the load-displacement-control (LDC) method using initial geometric imperfections will be tried and the implementation of the 2nd buckling mode shape as imperfection as well. In addition, the compression flange will be regarded as a column subjected to a combination of an axial load and a transverse load, caused by twisting of the beam.

## 1.4 Limitations

The cross sections used in the current thesis are uniform over the beam length and will be limited to: UPE160 section for channel beams, adjusted for modelling with shell elements, and the monosymmetric I-beam will have a cross section with the height $300$ mm, the width of the compression flange $150$ mm and the width of the tension flange $75$ mm (more details can be found in Section 4.1). The beams will be modelled as linearly elastic, steel beams with the Young's modulus of elasticity $E = 210$ GPa. The beams are singly spanned, unbraced with hinged supports. The length of the monosymmetric I-beam is $8.0$ m and the channel beams are $2.8$ m and $4.0$ m.

The monosymmetric I-beam will be subjected to a concentrated load, acting at different heights in the section, i.e. its shear centre, centre of gravity, top and bottom of the web. The different lateral restraints for the monosymmetric I-beam are limited to the degree of end-restraint for rotation about the minor axis and warping. The centrically loaded channel beams will be subjected to concentrated load at mid-span and uniformly distributed load over the whole span. For the eccentrically loaded channel beam, the eccentricity of the two types of loads is set to $5$ mm, $10$ mm and $26.8$ mm respectively. The support conditions will be applied at the shear centre of the end sections for all studied cases, if possible.

The commercial structural engineering software used in the present thesis are ADINA, Colbeam, LTBeam, SAP2000 and STAAD.Pro. ADINA and LTBeam use FEM modelling techniques in order to estimate $M_{cr}$, but the other use analytical expressions. The FEM analyses carried out in ADINA are a linearized buckling analysis and a collapse analysis with the LDC method using initial geometrical imperfections.

# 2 Theory

## 2.1 Beam modelling properties

### 2.1.1 Coordinate system

The *coordinate system* used hereafter is a Cartesian coordinate system where the x-axis lies along the beam, the positive z-direction is vertical pointing upwards and the y-direction is horizontal, perpendicular to the x-axis, as used in Eurocode 3 (Höglund 2006, p. 1). The *origin*, noted as OP in Figure 2.1, is located at the centroid (see Section 2.1.4).

For a vertically loaded beam the y-axis is considered its *major axis* and the z-axis its *minor axis*. Bending about the y-axis will consequently be called *major axis bending* and bending about the z-axis *minor axis bending*.

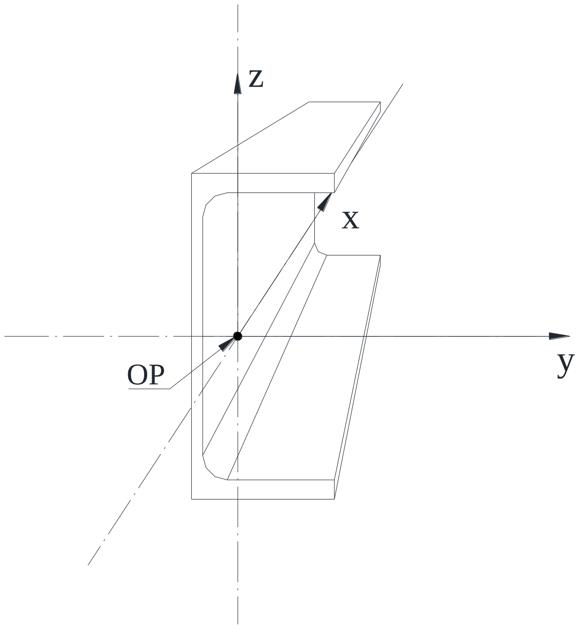



*Figure 2.1 Coordinate system for a channel cross section with the origin noted as OP.*

### 2.1.2 Degrees of freedom

A general beam element has two nodes, see Figure 2.2. Each node has 6 degrees of freedom; *translations* in the x-, y- and z-direction and *rotations* about the x-, y- and z-axis. A special kind of a beam element, hereafter referred to as a warping beam element, has an additional *warping* degree of freedom, to take warping into account (see Section 2.1.10).

Shell elements can vary in shape, have different number of nodes and integration points, for example. Each node in a shell element is considered to have the same 6 degrees of freedom as a general beam element; 3 translational and 3 rotational.

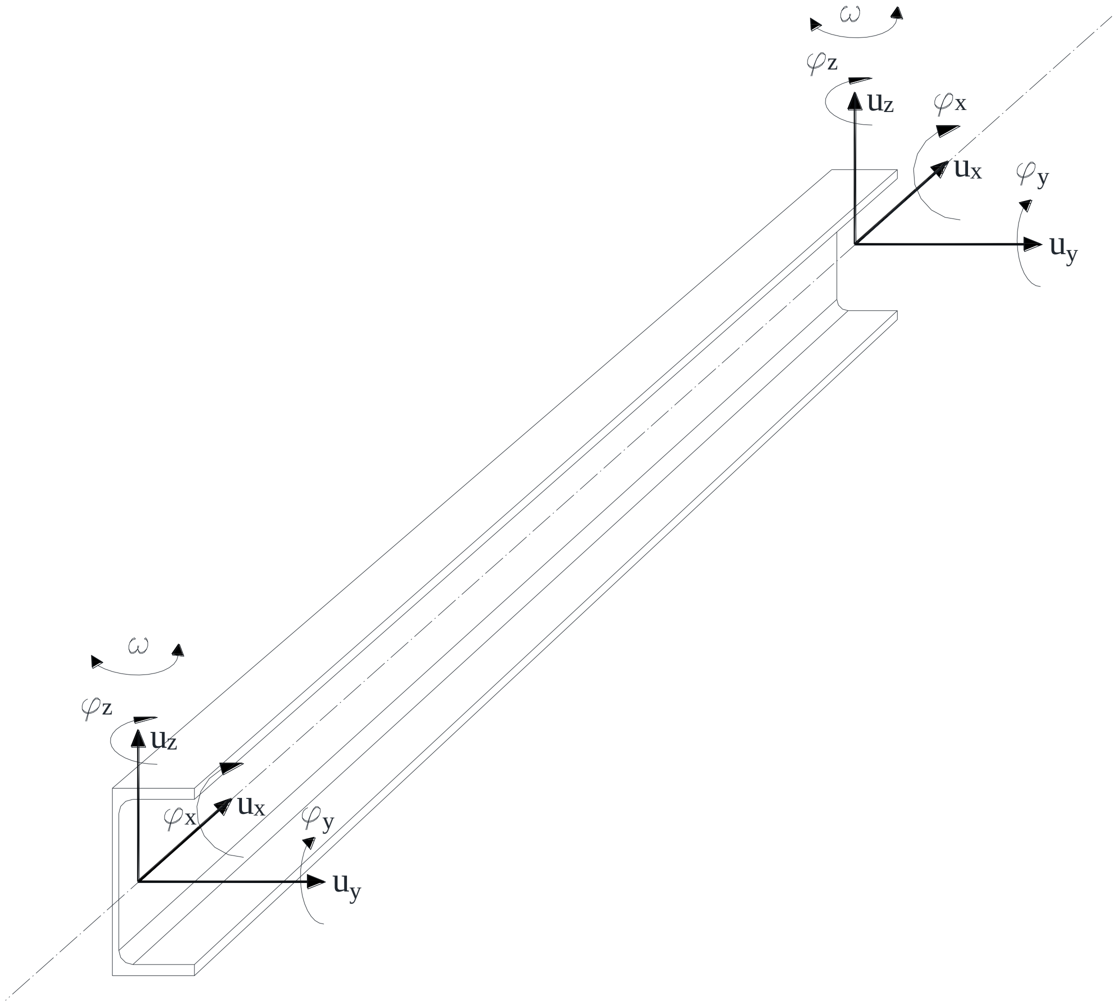



*Figure 2.2 Degrees of freedom for a warping beam element. Three translational degrees of freedom in the x-, y- and z-directions noted with $u_x$, $u_y$ and $u_z$ respectively, 3 rotational degrees of freedom about the x-, y- and z-axis noted with $\varphi_x$, $\varphi_y$ and $\varphi_z$ respectively and a warping degree of freedom noted with $\omega$.*

### 2.1.3 Centre of gravity

The c*entre of gravity*, GC, is the origin of the coordinate system. When looking at a cross section of a homogeneous beam, this point is where the weighted relative position of the mass sums to zero (Lundh 2013, p. 80). See Figure 2.3 for the position of the GC for a channel section, a monosymmetric I-section and a doubly symmetric I-section.

### 2.1.4 Centroid

For beams with a cross section of a homogeneous material, the *centroid* coincides with the GC as the density of the material is not taken into account, only the geometry.

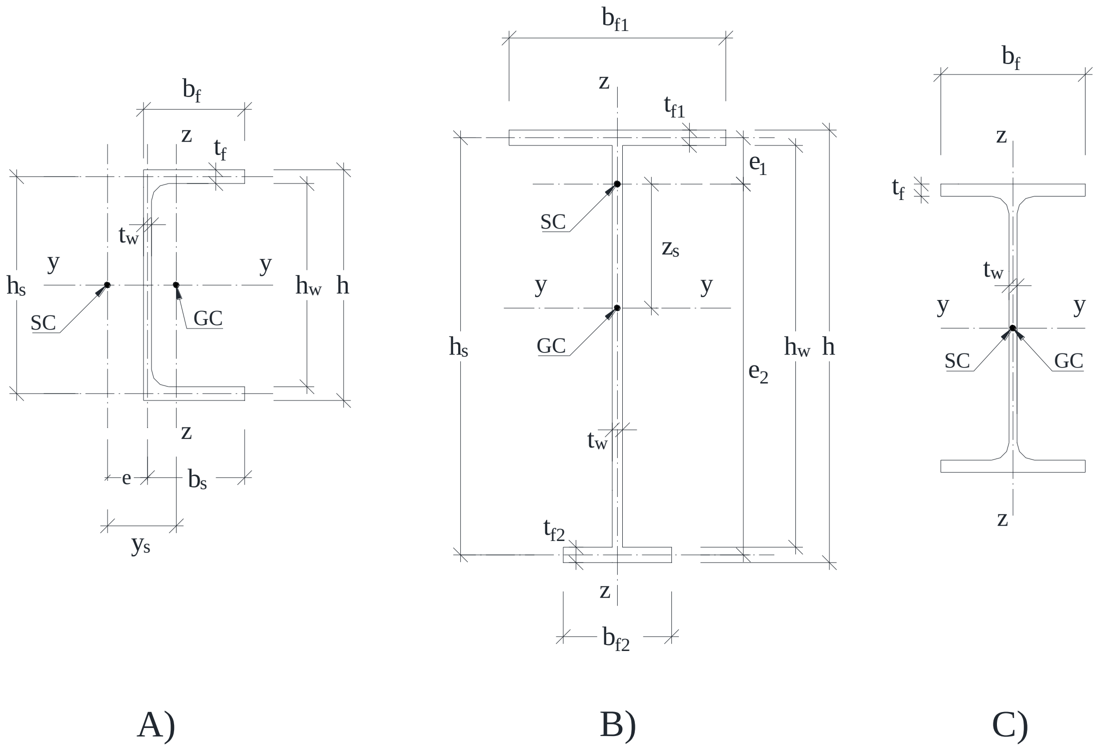



*Figure 2.3 Labelling of cross section properties for a channel section (A), a monosymmetric I-section (B) and a doubly symmetric I-section (C).*

### 2.1.5 Shear centre

The *shear centre*, SC, of a cross section is the point where pure bending of the beam occurs when loaded in the SC. See Figure 2.3 for the position of the SC for a channel section, a monosymmetric I-section and a doubly symmetric I-section.

For doubly symmetric I-beams, the SC coincides with the GC. That can be seen from equations (2.1) and (2.2) where $I_{f1} = I_{f2}$ which leads to $e_1 = e_2$.

For monosymmetric I-sections, the SC is located in the line of symmetry. (Höglund 2006, p. 29) The position on the symmetry line (z-axis), see Figure 2.3, is determined by the second moments of area of the flanges about the z-axis:

$$e_1 = \frac{I_{f2}}{I_z} h_s \tag{2.1}$$

$$\mathrm{e}_2 = \frac{I_{f1}}{I_z} h_s \tag{2.2}$$

| | | | |
|---|---|---|---|
| Where | $e_1$ | = | Distance between the top flange and the shear centre of the section |
| | $e_2$ | = | Distance between the bottom flange and the shear centre of the section |

| | | |
|---|---|---|
| $h_s$ | = | Distance between the shear centre of the top flange and the shear centre of the bottom flange of a beam |
| $I_{f1}$ | = | Second moment of area for the top flange about the minor axis |
| $I_{f2}$ | = | Second moment of area for the bottom flange about the minor axis |
| $I_z$ | = | Second moment of area about the minor axis |

When a beam with a channel section is loaded in the GC, a torsional effect occurs in addition to the bending. This is called the Bachian anomaly and results in twisting of the beam when loaded (Höglund 2006, p. 30).

Location of the SC of a channel section is on the opposite side of the web to the flanges. The distance $e$ from the SC to the centre line of the web, see the channel section on Figure 2.3, is given by the following equation (Höglund 2006, p. 30).

$$e = \frac{b_s^2 h_s^2 t_f}{4I_y} \tag{2.3}$$

| | | | |
|---|---|---|---|
| Where | $e$ | = | Distance from SC to the centre of the web |
| | $b_s$ | = | Width of the flanges to the centre of the web |
| | $t_f$ | = | Thickness of the beam flanges |
| | $I_y$ | = | Second moment of area about the major axis |

Different expressions exist for calculating the distance $e$ for channel sections, such as (Hoogenboom 2006):

$$e = \frac{3b_f^2 t_f}{6b_f t_f + h_s t_w} \tag{2.4}$$

| | | | |
|---|---|---|---|
| Where | $b_f$ | = | Width of the beam |
| | $t_w$ | = | Thickness of the web |

And also (CISC 2002):

$$e = \frac{b_s}{2 + \frac{h_s t_w}{3b_s t_f}} \tag{2.5}$$

By comparing equations (2.4) and (2.5), it can be seen that the only difference is which width parameter is used, $b_f$ or $b_s$. In Höglund (2006) it is not noted clearly which parameter should be used in equation (2.3). In Section 2.1.9 some different methods for considering the length of cross section parts will be discussed further.

The variables $z_s$ and $y_s$ are the coordinates of the SC with respect to the GC in the z- and the y-directions respectively, as showed in Figure 2.3.

### 2.1.6 Torsion centre

If a beam with an open section is subjected to torsion, every cross section will rotate around the *torsion centre*, TC, if warping is prevented in any section. The TC coincides with the SC (Höglund 2006, p. 28).

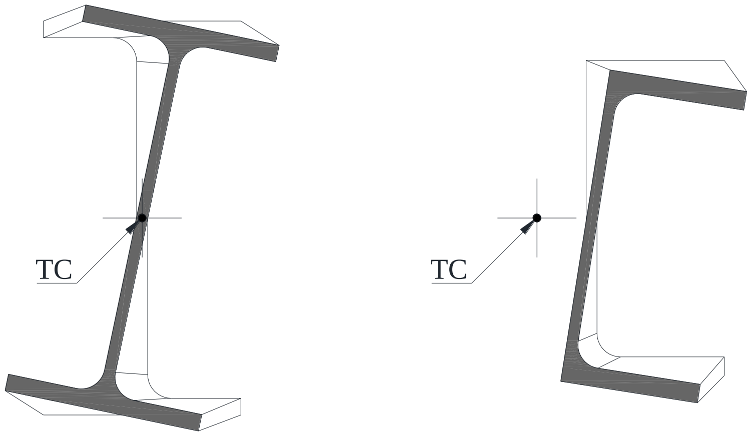



*Figure 2.4 Position of the torsion centre for an I-section and a channel section.*

### 2.1.7 Point of load application

The *point of load application*, PLA, represents the point where the load acts on the cross section of a beam, see examples in Figure 2.5. The term $z_g$ is usually used to describe the coordinate of the PLA with respect to the SC in the z-direction. Another common term is $z_a$, which is the coordinate of the PLA with respect to the GC in the z-direction. Respectively the location of the PLA in the y-direction can be described by the terms $y_g$ and $y_a$. $y_g = 0$ represents a *centric* loading and $y_g \neq 0$ an *eccentric* loading. For beams with a cross section symmetric about the y-axis, $z_g$ and $z_a$ are equal. For beams with a cross section symmetric about the z-axis, such as I-beams, $y_g$ and $y_a$ are equal.

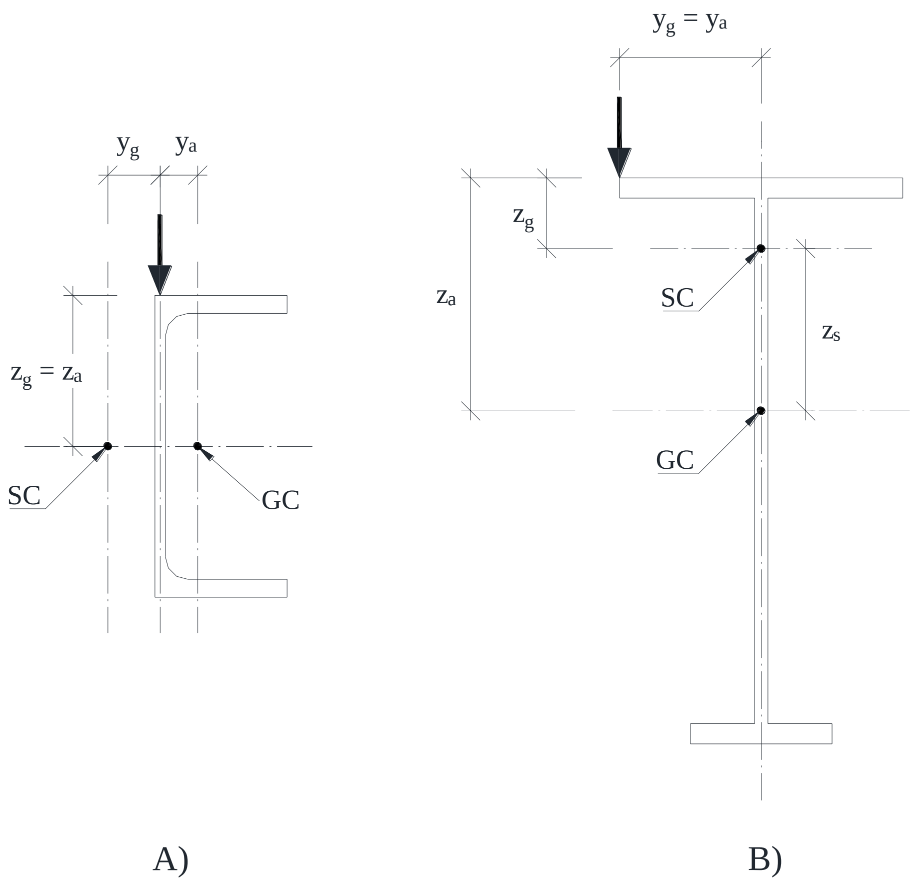



*Figure 2.5* *A point load applied with at a distance $y_g$ and $z_g$ from the shear centre on a channel section (A) and on a monosymmetric I-section (B).*

### 2.1.8 Support conditions

In order to have a system that is in equilibrium, a model of a loaded beam must include supports, i.e. some degrees of freedom have to be restrained. Looking at the ordinary single span beam, it is supported at both ends and regardless of the support type; the y- and z-translations are restrained. The translation along the beam, in the x-direction, must be restrained at least at one node and most commonly the x-translation is fixed at one end but free at the other. To prevent the beam from rotating about its own longitudinal axis, the x-rotation should be restrained at both supports.

Consider the single spanned beam, with supports at both ends, modelled with the warping beam elements. The 3 of the end nodes' 7 degrees of freedom that are left and available for altering are:

- Rotation about y-axis ($k_y$)
- Rotation about z-axis ($k_z$)
- Warping ($k_w$)

Each of them can be fixed, free or even something in between and they all influence the global behaviour of the beam when it is loaded. These 3 degrees of freedom are often related to the effective buckling length, by assigning the effective buckling length factors $k_y$, $k_z$ and $k_w$ for each of them like shown in the list above, that applies to the beam as a whole. In general the value of an effective buckling length factor varies from 0.5 for full restraint to 1.0 for no restraint and takes the value 0.7 for a beam fixed at one end and free at the other (ECCS 2006, p. 229).


*Figure 2.6* *Two common single spanned beam models. Model A) is considered to have fixed supports, where rotations about the y-axis and the z-axis as well as warping are all fixed at both supports so that* $k_y = k_z = k_w = 0.5$. *Model B) is considered to have hinged supports, where rotations about the y-axis and the z-axis as well as warping are all free at both supports so that* $k_y = k_z = k_w = 1.0$.

Two of the most common single spanned beam models are introduced in Figure 2.6 as examples. For a beam with the y-rotation, z-rotation and warping fixed at both ends, all three buckling length factors are equal to 0.5. Such a beam, see Figure 2.6 A), is considered to have fixed supports. Another case is when the y-rotation, z-rotation and warping are all free at both ends. For that beam, see Figure 2.6 B), all the buckling length factors are equal to 1.0 and it would be thought of as a beam with hinged supports. The supports of the latter beam are more specifically called *fork supports*, see illustration in Figure 2.7.

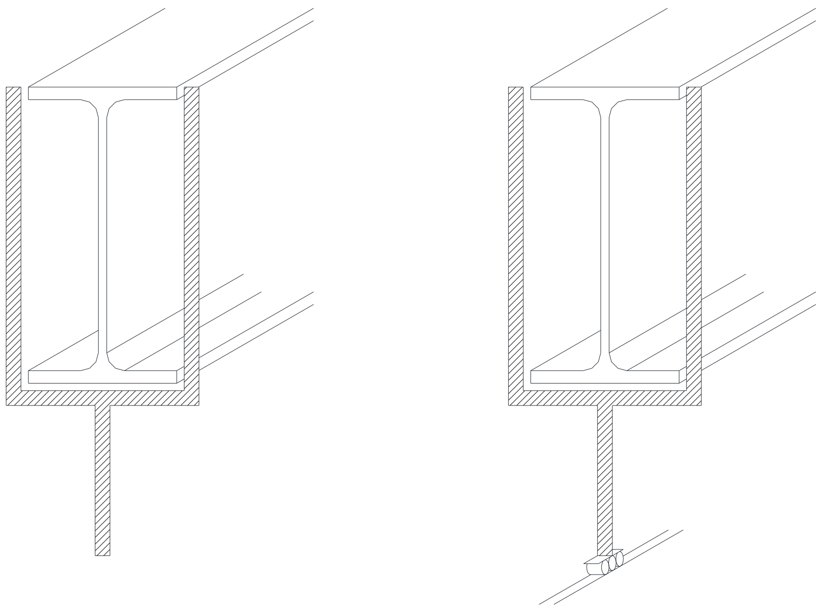



*Figure 2.7* *Illustration of fork supports for an I-beam. The rotation about x-axis is fixed and the beam is free to rotate about the minor axis and the major axis. In end support A) translations in all directions are fixed, but at the end B) y- and z-translations are prevented and x-translation is free.*

### 2.1.9 Torsion

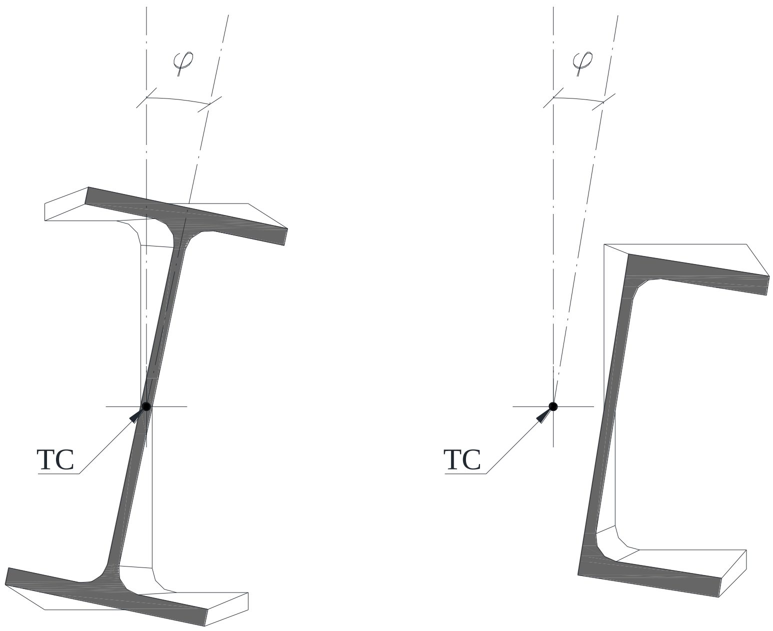



*Figure 2.8* *Twisting of I-shaped and channel cross sections with φ as the angle of twist.*

According to the Saint-Venant torsion theory the typical torsion stresses occur only when free warping (see Section 2.1.10) can take place. That situation is not common in practise. Another restriction for Saint-Venant torsion is that distributed moment loading is not allowed (Hoogenboom 2006). The basic equation for torsion is according to Saint-Venant is as follows:

$$\frac{d\varphi}{dx} = \frac{T}{GI_t} \tag{2.6}$$

| | | | |
|---|---|---|---|
| Where | $T$ | = | Torsion |
| | $\varphi$ | = | Angle of twist (see Figure 2.8) |
| | $G$ | = | Shear modulus |
| | $I_t$ | = | Saint-Venant torsional constant |
| | $GI_t$ | = | Torsional stiffness |

This is also called *Saint-Venant torsion* (Höglund 2006, p. 31).

V. Z. Vlasov developed a theory to take restraint warping into account. The theory is called *warping torsion* or *non-uniform torsion*. The theory includes a non-uniform torsion along a beam. The rotation of the cross section along a beam can be described by the following differential equation:

$$EI_w \frac{d^4\varphi}{dx^4} - GI_t \frac{d^2\varphi}{dx^2} = m_x \tag{2.7}$$

| | | | |
|---|---|---|---|
| Where | $E$ | = | Young’s modulus of elasticity |
| | $I_w$ | = | Warping constant |
| | $EI_w$ | = | Warping stiffness |
| | $m_x$ | = | Distributed torsional moment along the beam |

For a cross section built up by rectangular parts, the torsional constant, often related to Saint-Venant’s theory, is most commonly expressed as (Lundh 2013, p. 338):

$$I_t = \sum_{i=1}^{n} \frac{t_i h_i^3}{3} \tag{2.8}$$

| | | | |
|---|---|---|---|
| Where | $t_i$ | = | Thickness of cross section part $i$ |
| | $h_i$ | = | Length of cross section part $i$ |
| | $n$ | = | Number of cross section parts |

Here the possible fillets in the cross section are not taken into account. For beams with an I-section or a channel section, this expression becomes (Access Steel 2006):

$$I_t = \frac{b_{f1}t_{f1}^3 + b_{f2}t_{f2}^3 + h_w t_w^3}{3} \tag{2.9}$$

| | | | |
|---|---|---|---|
| Where | $b_{f1}$ | = | Width of the top flange |
| | $t_{f1}$ | = | Thickness of the top flange |
| | $b_{f2}$ | = | Width of the bottom flange |
| | $t_{f2}$ | = | Thickness of the bottom flange |
| | $h_w$ | = | Height of the web |
| | $t_w$ | = | Thickness of the web |

The expression for the torsional constant above is an approximation and there are other versions of the expression to be found in other literature. In equation (2.9) the length of the cross section parts is taken as their net length, as can be seen in case A) in Figure 2.9, which is considered to be a rather conservative approach (CISC 2002).

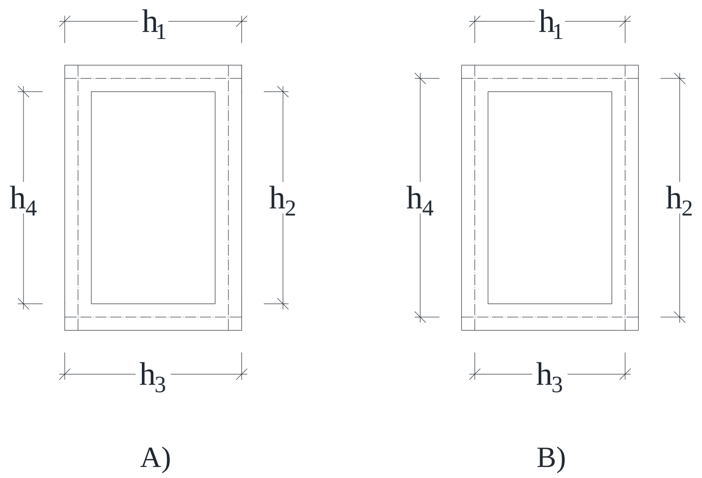



*Figure 2.9* *An illustrative comparison of two methods to measure the length of cross section parts (see $h_i$ in equation (2.8). In A) the length is taken as the net length of the cross section parts. In B) the length is taken between the points of intersection of the local longitudinal axes of the cross section parts.*

Often the length of the cross section parts, mentioned in equation (2.8), is considered to be between the points of intersection of their local longitudinal axes, as illustrated in case B) in Figure 2.9 (Galambos 1968). In that case the expression for the torsional constant for an I-section becomes (CISC 2002 & Hoogenboom 2006):

$$I_t = \frac{b_{f1}t_{f1}^3 + b_{f2}t_{f2}^3 + h_s t_w^3}{3} \tag{2.10}$$

Where $h_s$ = Distance between the shear centre of the top flange and the shear centre of the bottom flange of a beam

And for a channel section with equal top and bottom flanges the torsional constant is then either expressed as (Hoogenboom 2006):

$$I_t = \frac{2b_f t_f^3 + h_s t_w^3}{3} \tag{2.11}$$

Where $b_f$ = Width of the beam if the top and bottom flanges are equal

$t_f$ = Thickness of the beam flanges if the top and bottom flanges are equal

Or alternatively (CISC 2002):

$$I_t = \frac{2b_s t_f^3 + h_s t_w^3}{3} \tag{2.12}$$

Where $b_s$ = Width of the flanges to the centre of the web

Other expressions for the torsional constant for I-sections include an additional multiplier such as (Höglund 2006, p. 33):

$$I_t = c \cdot \frac{b_{f1}t_{f1}^3 + b_{f2}t_{f2}^3 + h_w t_w^3}{3} \tag{2.13}$$

Where $c$ = Torsional constant multiplier

It is stated that the torsional constant multiplier $c$ varies from 1.1 to 1.2 depending on the cross section shape, and often taken as 1.15, without further clarification (Höglund 2006, p. 33).

Anther expression for the torsional constant of I-sections (Ziemian 2010):

$$I_t = \frac{b_{f1}t_{f1}^3}{3}\left(1 - 0.63\frac{t_{f1}}{b_{f1}}\right) + \frac{b_{f2}t_{f2}^3}{3}\left(1 - 0.63\frac{t_{f2}}{b_{f2}}\right) + \frac{h_w t_w^3}{3} \tag{2.14}$$

### 2.1.10 Warping

When torsion is applied to a beam, every section will rotate around its TC. For circular members every section rotates in plane without any out of plane deformation. For members with other cross sections, the sections rotate as in the case of members with circular cross sections, but in addition to that the cross section twists about the z- and/or the y-axis which results in *warping*, see Figure 2.10.

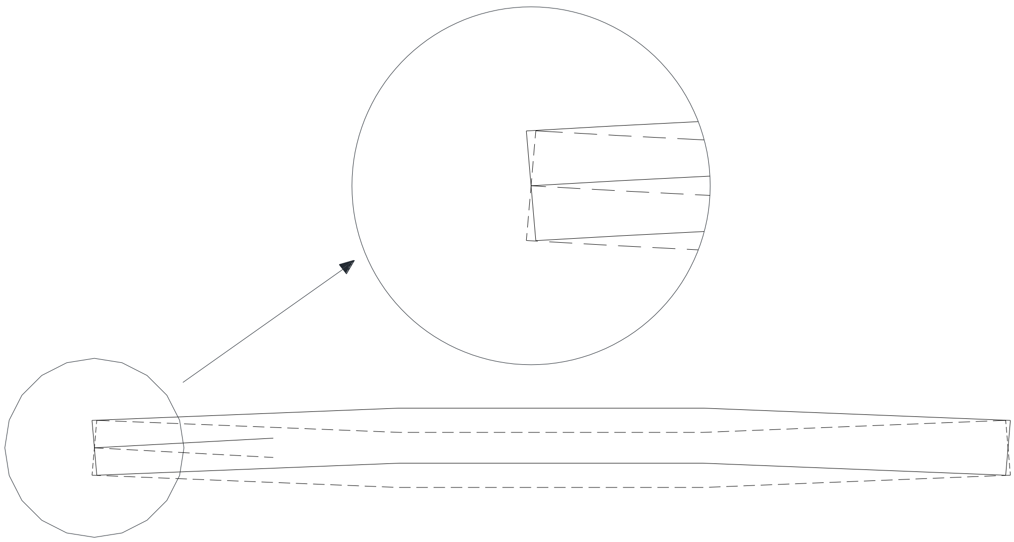


*Figure 2.10 Illustration of a beam that is free to warp at its end supports seen from above. The top flange is expressed with solid lines and the bottom flange with dashed lines.*

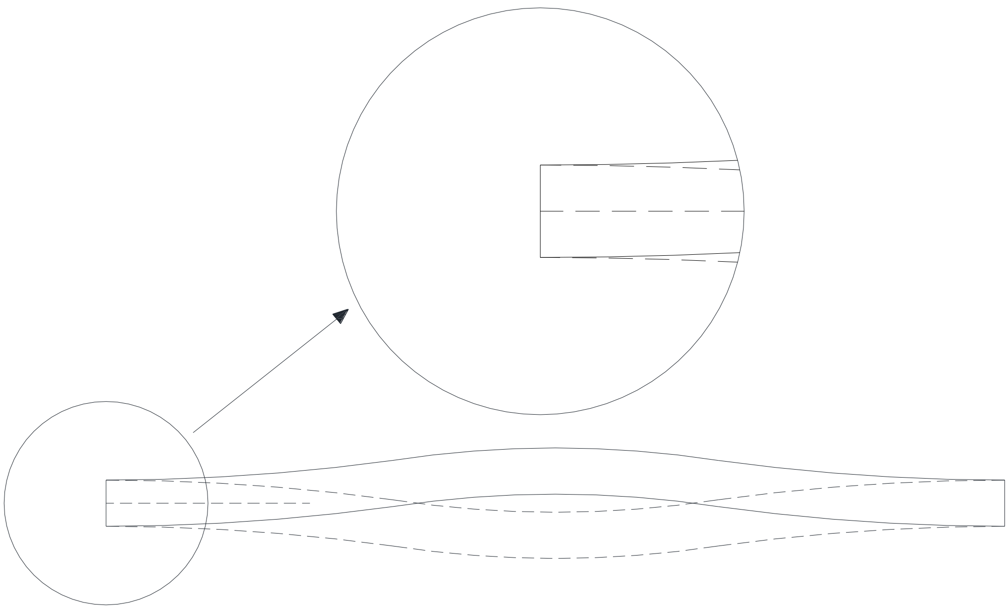


*Figure 2.11 Illustration of a beam with warping restrained at its end supports seen from above. The top flange is expressed with solid lines and the bottom flange with dashed lines.*

Warping of a beam can be restrained, see Figure 2.11, for example at the supports if the beam is welded to a thick plate. Also, warping is restrained where an imposed torsional moment is applied (Hoogenboom 2006).

The warping constant is defined as:

$$I_w = \int_A \psi^2 dA \tag{2.15}$$

| | | | |
|---|---|---|---|
| Where | $\psi$ | = | Warping function |
| | $I_w$ | = | Warping constant |

Just like the torsional constant, see Section 2.1.9, the warping constant is usually approximated and possible fillets disregarded, and different kinds of expressions exist in different literature. According to ECCS (2006) the warping constant can be calculated as follows for I-sections:

$$I_w = \left(1 - \psi_f^2\right) I_z \left(\frac{h_s}{2}\right)^2 \tag{2.16}$$

| | | | |
|---|---|---|---|
| Where | $\psi_f$ | = | Differential ratio of the second moments of area about the minor axis for the beam flanges |
| | $I_z$ | = | Second moment of area about the minor axis |
| | $h_s$ | = | Distance between the shear centre of the top flange and the shear centre of the bottom flange of a beam |

The ratio is calculated with the following formula and is only nonzero for monosymmetric I-sections.

$$\psi_f = \frac{I_{f1} - I_{f2}}{I_{f1} + I_{f2}} \tag{2.17}$$

| | | | |
|---|---|---|---|
| Where | $I_{f1}$ | = | Second moment of area for the top flange about the minor axis |
| | $I_{f2}$ | = | Second moment of area for the bottom flange about the minor axis |

The same formula for the warping constant for I-sections as equation (2.16) can be found but expressed differently as (Access Steel 2006):

$$I_w = h_s^2 I_z \frac{b_{f1}^3 t_{f1} b_{f2}^3 t_{f2}}{\left(b_{f1}^3 t_{f1} + b_{f2}^3 t_{f2}\right)^2} \tag{2.18}$$

| | | | |
|---|---|---|---|
| Where | $b_{f1}$ | = | Width of the top flange |

| | | |
|---|---|---|
| $b_{f2}$ | = | Width of the bottom flange |
| $t_{f1}$ | = | Thickness of the top flange |
| $t_{f2}$ | = | Thickness of the bottom flange |

If the length of the cross section parts is considered to be between the points of intersection on their longitudinal axis, like discussed in Section 2.1.9, the warping constant is formulated differently. For an I-section the expression becomes (CISC 2002 & Ziemian 2010):

$$I_w = \frac{h_s^2 b_{f1}^3 t_{f1}}{12\left(1 + \left(\frac{b_{f1}}{b_{f2}}\right)^3 \left(\frac{t_{f1}}{t_{f2}}\right)\right)} \tag{2.19}$$

For channel sections, where the top and bottom flanges are assumed to be equal and the length of the cross section parts is considered to be between the intersection points on their longitudinal axis, the warping constant can be calculated as follows (CISC 2002):

$$I_w = h_s^2 b_s^3 t_f \left(\frac{1-3a}{6} + \frac{a^2}{2}\left(1 + \frac{h_s t_w}{6 b_s t_f}\right)\right) \tag{2.20}$$

| | | | |
|---|---|---|---|
| Where | $a$ | = | Value to determine the warping constant for channel sections |
| | $t_w$ | = | Thickness of the web |

The term $a$ is calculated as:

$$a = \frac{1}{2 + \frac{h_s t_w}{3 b_s t_f}} \tag{2.21}$$

| | | | |
|---|---|---|---|
| Where | $b_s$ | = | Width of the flanges to the centre of the web |

Another expression for the warping constant of channel sections is (Hoogenboom 2006):

$$I_w = \frac{h_s^2 b_f^3 t_f}{12} \frac{3 b_f t_f + 2 h_s t_w}{6 b_f t_f + h_s t_w} \tag{2.22}$$

| | | | |
|---|---|---|---|
| Where | $b_f$ | = | Width of the beam if the top and bottom flanges are equal |
| | $t_f$ | = | Thickness of the beam flanges |

Equation (2.22) gives the same result as equation (2.20) if the width $b_s$ is used instead of $b_f$, i.e. if the length of the flanges is considered to be to the point of intersection with the web on their longitudinal axis.

A *bi-moment* is a term related to warping and occurs when warping is restrained. It can be described as the distribution of axial stresses needed to prevent the warping. For I-beams it can be described as the moment in each of the flanges times the distance between them. For other sections, this is more complicated (Hoogenboom 2006). The definition of the bi-moment is:

$$B = -\int_A \sigma_{xx}\psi dA \qquad (2.23)$$

Where $\sigma_{xx}$ = Axial stress in the x-direction

$B$ = Bi-moment

For a beam with an I-section the bi-moment can be expressed as (Hoogenboom 2006):

$$B = M_{fl}h_s \qquad (2.24)$$

Where $M_{fl}$ = Moment in the flanges, see Figure 2.12

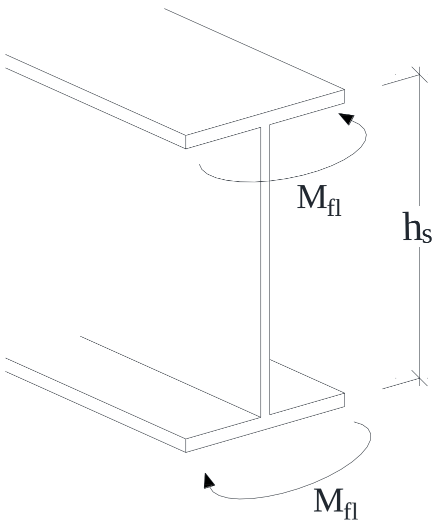



*Figure 2.12 Illustration of how a bi-moment can be interpreted for an I-section (adapted from Hoogenboom 2006).*

## 2.2 Buckling

Buckling is an instability phenomenon caused by a bifurcation of equilibrium and is best described by considering different states of equilibrium. It implies that there is a load level where further loading can lead to more than one equilibrium state (see Figure 2.14).

### 2.2.1 Stability

A theory called *the classical stability theory* is based on three main assumptions (Höglund 2006, p. 3):

- Linear elastic material
- Ideal shape of structural elements
- Small displacements assumed for constitutive relations

In the theory it is differentiated between three states of equilibrium; *stable*, *indifferent* and *unstable*, illustrated with cases A), B) and C) in Figure 2.13 respectively. In the stable state of equilibrium a structure goes back to its original state after a small disturbance. When a structure is in an unstable equilibrium, only a small disturbance is needed to give rise to forces that take the structure further away from the original state. If a disturbance leads to a change in the structure that neither goes back nor increases after the disturbance ceases, it is referred to as an indifferent state of equilibrium (Höglund 2006, p. 2)

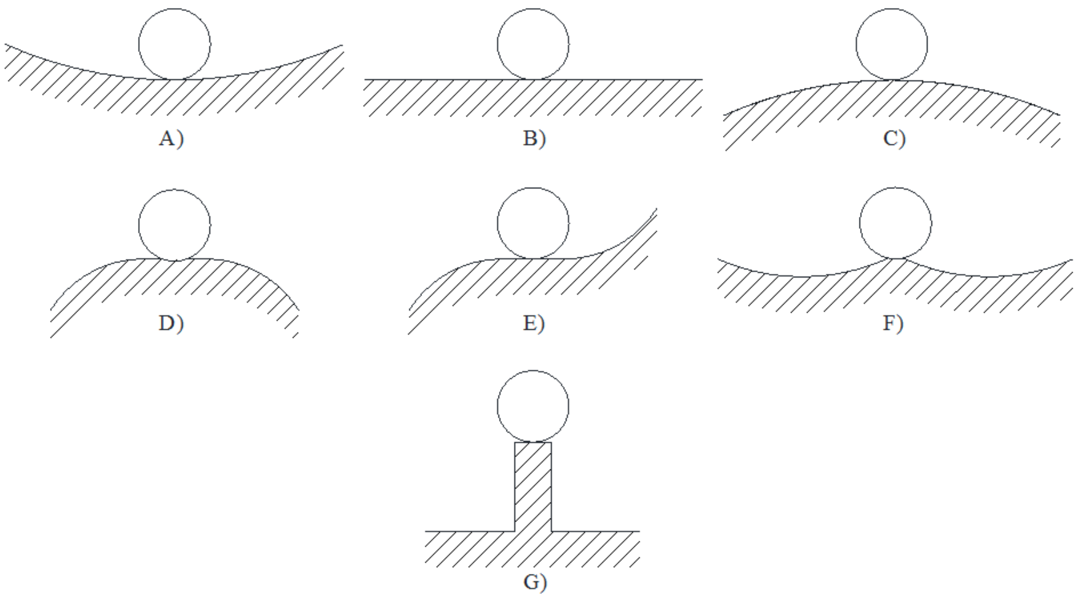



*Figure 2.13 Illustration of different states of equilibrium (adapted from Höglund 2006, p. 3).*

More types of equilibrium states can be thought of and some of them are presented in Figure 2.13. Case D) shows a system that is stable as long as it is only affected by a very small disturbance, but as soon as the disturbance is big enough the system becomes unstable. The effect of a disturbance can be dependent on the direction or the type of the disturbance in the same way as the magnitude, as case E) represents. Equilibrium case F) illustrates a structure that, due to a small disturbance, becomes more stable after being transferred from its initial state. It can be said that the structure, for instance a thin sheet, has a post-critical strength. An equilibrium state that is stable but highly dependent on the effects of a small disturbance is shown in case G), which is quite common for cases with shell buckling that have initial imperfections or load eccentricities as disturbances (Höglund 2006, p. 3).

Considering a beam with a bending moment acting about its major axis, its state of equilibrium can be examined by adding and removing a small lateral force in order to disturb the beams equilibrium. Should the beam return to its original shape after the disturbance is removed, its straight state of equilibrium is stable. If the moment is increased to a value equal to or larger than a certain critical value (described further in Section 2.2.2), the beam will not return to a straight shape and its equilibrium state is then unstable. In theory an undisturbed beam can be loaded beyond the critical value without lateral deformation, but as soon as a disturbance is made the lateral deformation appears. For a linear elastic beam under all loading levels there is a stable equilibrium state, as illustrated in Figure 2.14. In the laterally deformed equilibrium state the deformation increases rapidly with further loading. The magnitude of the lateral deformation cannot be solved using the classical theory, since it assumes small deformation as mentioned above, so the *theory for large deformations* is needed (Höglund 2006, p. 11).

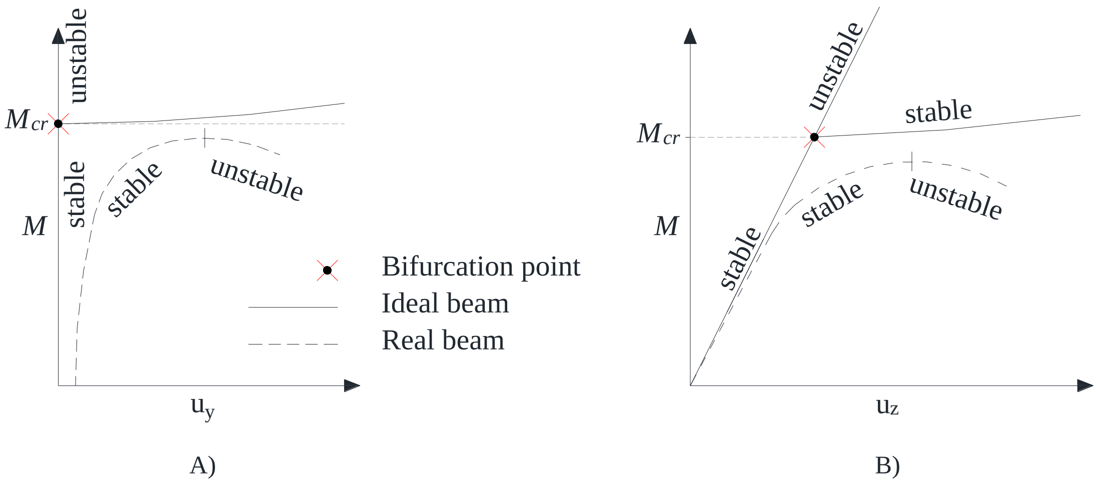



*Figure 2.14 Relation between an applied load and a centre span displacement for a beam loaded with a bending moment acting about its major axis. In A) the load is plotted versus the lateral displacement $u_y$ and in B) the load is plotted versus the vertical displacement $u_z$. A comparison is also made between the behaviour of ideal beams and real beams in lateral-torsional buckling, and the states of equilibrium noted (adapted from Höglund 2006, p. 11 & p. 82).*

### 2.2.2 Behaviour of ideal beams and real beams in bending

In theory only a perfect geometry under elastic conditions is considered, but in reality that is never the case and it results in different behaviour than predicted. Here only lateral-torsional buckling behaviour will be considered (see Section 2.2.5.4).

An *ideal beam* is a beam that is free of imperfections, such as initial bow imperfection and residual stresses, and made of linearly elastic material. When a bending moment about the major axis is applied on such a beam its behaviour in lateral-torsional buckling can be described as follows. The beam only deforms vertically for loads up to a certain magnitude called the *critical load*. At that point the beam goes from a stable equilibrium position to an indifferent one (see Section 2.2.1) and a sudden lateral deflection and a twist occur simultaneously. A small increase of the load, beyond the critical load, results in a significant additional deformation. The beams state of equilibrium can be considered stable since its material is linearly elastic. A decrease of the load below the critical value however brings the beam back to a perfectly vertical deflection, with no lateral deflection or twist (Höglund 2006, p. 81).

A *real beam* always has some imperfections. The initial lateral bow of the beam results in an immediate increase in lateral deflection and a twist as the beam is loaded. When the load gets close to the critical load the lateral deflection and the twist increases more rapidly, but the theoretical critical load can never be reached (Höglund 2006, p. 81). The five factors that affect the lateral-torsional buckling of a real beam are (Höglund 2006, p. 83):

- Non-linear material properties (plastic response)
- Amplitude and shape of the initial lateral imperfections
- Residual stresses due to a shrinkage of the steel during manufacture
- Local buckling for the parts in cross section class 4
- Defects such as holes and asymmetry

The different behaviour of ideal beams and real beams is illustrated in Figure 2.14.

### 2.2.3 Local buckling

Buckling may be governed by *local buckling* for beams with slender webs or flanges. It can be recognized by many small buckles along the web or the flanges. For local buckling of the web, the buckling length is approximately the width of the web, see Figure 2.15. For a flange the buckling length can vary between 1 and 5 times the width of the flange (Höglund 2006, p. 1).

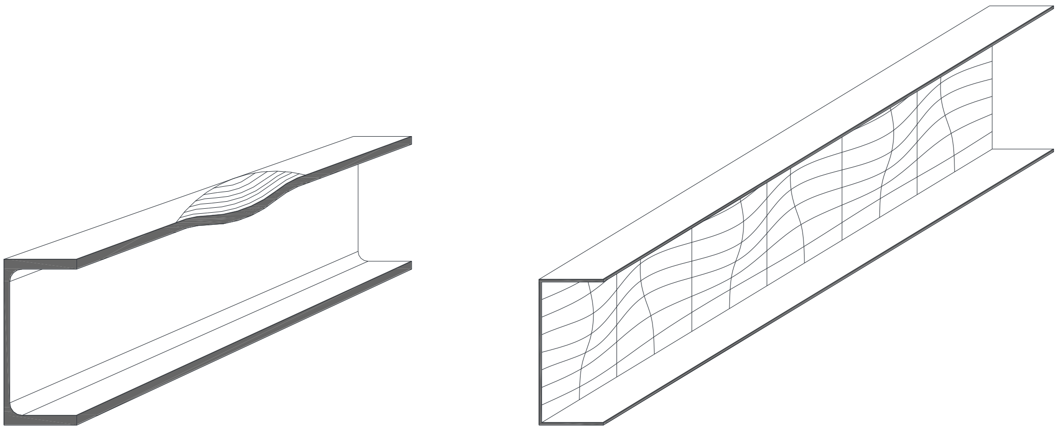


*Figure 2.15 Illustrations of local buckling of a flange (left) and a web (right).*

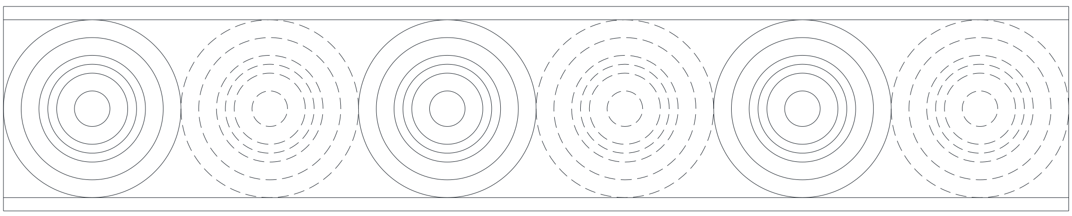


*Figure 2.16 A side-view of the local buckling of the web in Figure 2.15.*

### 2.2.4 Distortional buckling

*Distortional buckling* may occur for comparatively short beams (Kumar & Samanta 2005). This is a result of an interaction between two buckling modes; lateral-torsional buckling and local buckling, which both are commonly designed for seperately.

When distortional buckling occurs, the web distorts and the flanges twist and deflect laterally, see Figure 2.17. The consequence is a reduced torsional resistance (Kumar & Samanta 2005).

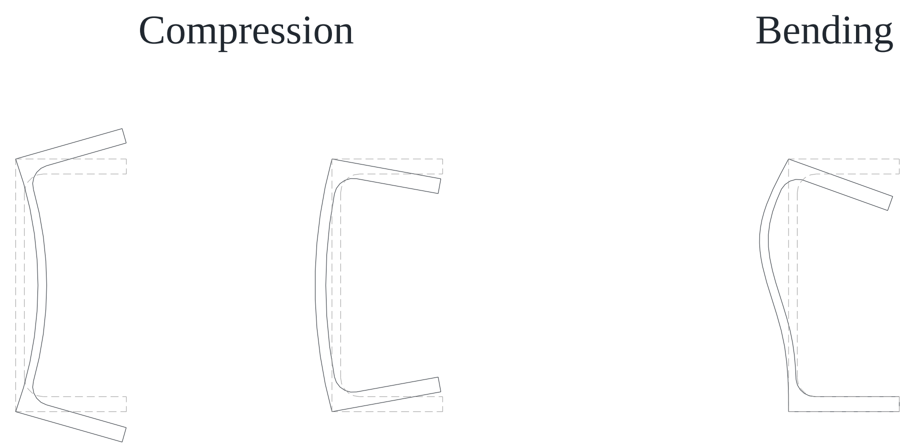



*Figure 2.17 Distortional buckling modes for a channel section (adapted from Hancock 1997 and Seah & Khong 1990). The dashed lines represent the original shape and the solid lines represent the shape after buckling.*

### 2.2.5 Global buckling

Buckling that occurs and influences the beam globally is called *global buckling*. This type of buckling can be subdivided further depending on what kind of deformation takes place and the type of load acting on the beam (Höglund 2006, p. 2).

#### 2.2.5.1 Flexural buckling

Under axial loading or simultaneous axial and moment loading, a beam may buckle in one plane without twisting. This is called *flexural buckling*, see Figure 2.18. If a member undergoes a pure axial load, it may buckle laterally and will take the shape of a sinus wave. The number of waves can vary and is for example dependent on the support conditions and lateral restraints (Höglund 2006, p. 13). These kinds of buckling are called the Euler's buckling cases.

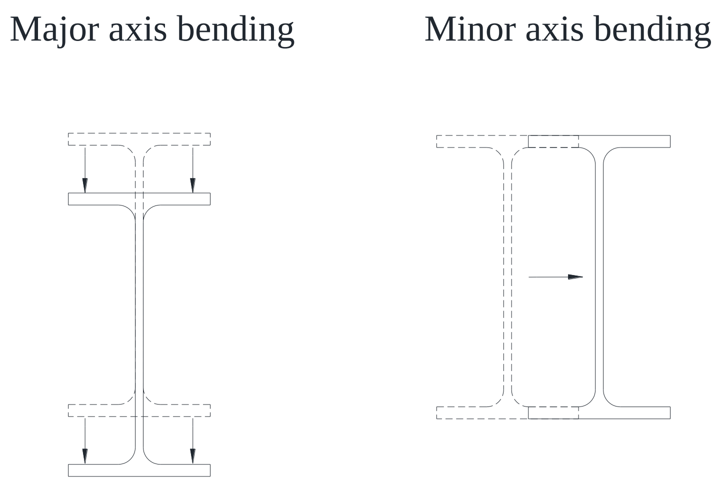



*Figure 2.18 Illustrations of flexural buckling due to a major axis bending (left) and a minor axis bending (right). The dashed lines represent the original shape and the solid lines represent the shape after buckling.*

### 2.2.5.2 Torsional buckling

Members with a small torsional stiffness can buckle in that way the cross section twists when an axial load is applied, as shown in Figure 2.19, (Höglund 2006, p. 2).

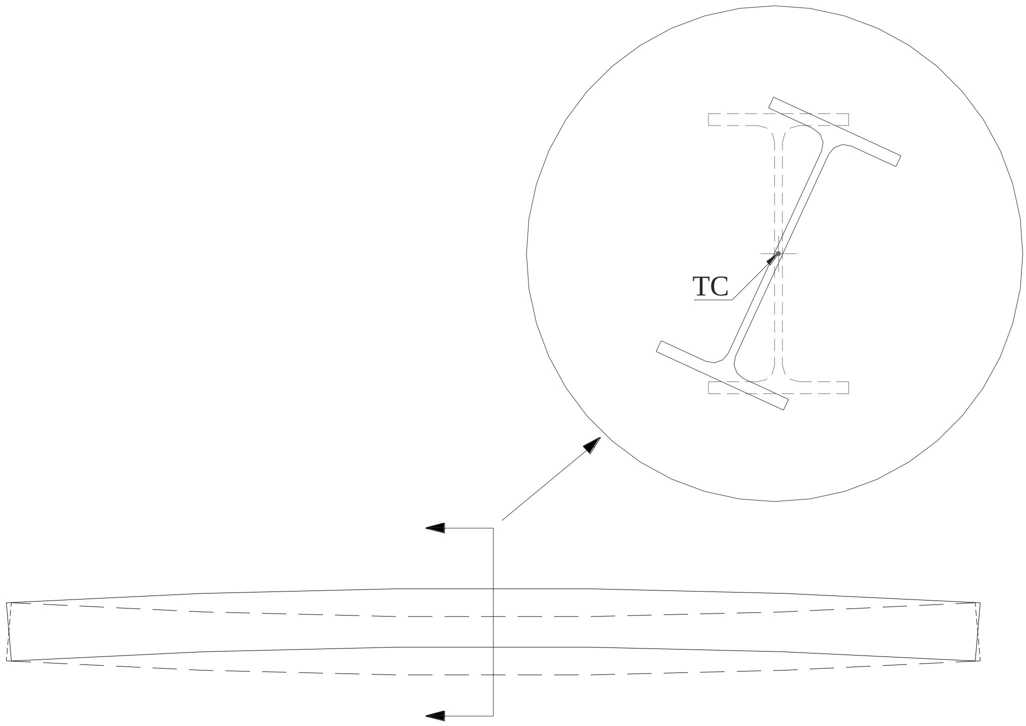



*Figure 2.19 Illustration of torsional buckling for an I-beam seen from above. The top flange is expressed with solid lines and the bottom flange with dashed lines. In the extracted cross section, the deformed shape is expressed with solid lines and the original shape with dashed lines.*

### 2.2.5.3 Torsional-flexural buckling

*Torsional-flexural buckling* may occur to axially loaded beams with a monosymmetric I-section and a channel cross section. The beam will simultaneously twist and deflect laterally when it buckles (Höglund 2006, p. 2). This can also occur to beams with doubly symmetric cross sections, in the cases where one flange is braced or a moment acts on the beam in addition to the axial load (Höglund 2006, p. 2).

### 2.2.5.4 Lateral-torsional buckling

When a pure moment acts on an I-beam about its major axis, one flange undergoes compression and the other tension. If the lateral stiffness is insufficient the compression flange will buckle laterally before the plastic (for class 1 and 2, elastic for 3 and effective for 4) moment resistance of the beam has been reached. The compression flange pushes the beam sideways which results in a lateral deflection. The tension flange pulls the beam towards its original location which results in twisting of the beam. This type of buckling is *lateral-torsional buckling,* see Figure 2.20. This may also occur when a simultaneous axial load and bending moment act on the beam. This applies for doubly symmetric and monosymmetric I-beams as well as beams with a channel section (Höglund 2006, p.2).

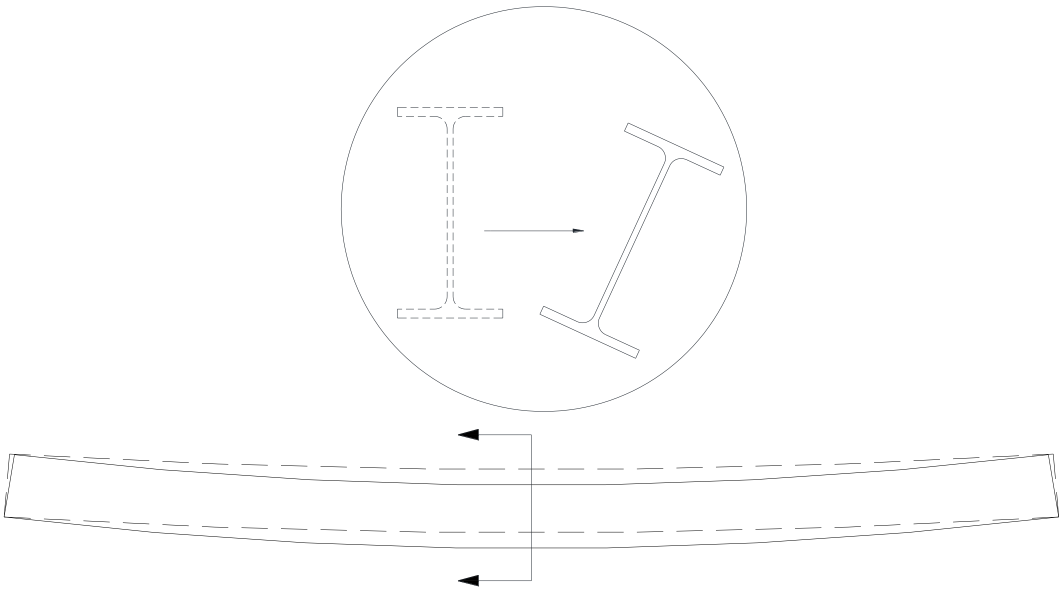


*Figure 2.20 Illustration of lateral-torsional buckling for an I-beam seen from above. The top flange is expressed with solid lines and the bottom flange with dashed lines. In the extracted cross section, the original shape is expressed with dashed lines and the deformed shape with solid lines.*

## 2.3 Energy methods

Energy relations and statements resulting from conditions for the potential energy of a system can be used to derive a theoretical critical buckling load. In generally accepted publications such as Timoshenko & Gere (1961), Chajes (1974) and Trahair (1993), for example, such energy methods have been used to establish solutions for simple models and load cases, which are the most common ones (Lopez et al. 2006).

The term $H$ refers to the *total potential energy* of a system and it can be separated into two parts, the *external potential energy* $H_e$ and the *internal potential energy* $H_i$. The external potential energy is due to the work performed by an external load. The work done by internal forces results in the internal potential energy and it is also equal to the elastic energy stored in the beam. What is most useful regarding the potential energy when studying buckling, is the *change in the potential energy* $\Delta H$ when a beam goes from a straight shape to a deflected shape of equilibrium (Höglund 2006, p. 36).

$$\Delta H = H_{deflected} - H_{straight} \tag{2.25}$$

| | | | |
|---|---|---|---|
| Where | $H_{deflected}$ | = | Total potential energy in the deflected equilibrium position |
| | $H_{straight}$ | = | Total potential energy in the straight equilibrium position immediately before buckling |

Here the total potential at the position where the beam goes from a straight shape right before buckling is considered to be a zero-level, so at that position $H_{straight} = 0$. Then the total potential in the deflected state can be thought of as the total potential of the system. The two following conditions apply for the total potential (Höglund 2006, p. 36 & p. 40):

- The total potential has a local minimum in the position of equilibrium. This refers to the deflected shape of a beam, i.e. if the right deflected shape is given, all changes to the shape will result in an increased total potential.
- The total potential doesn't change when a beam goes from the straight shape to the deflected shape of equilibrium. This means that at the buckling load the beam can change to the deflected shape without a change in the total potential.

From these two conditions the theoretical critical load can be derived with calculus of variations for simple cases, which leads to the elastic critical moment $M_{cr}$ (see Section 2.4), or approximated for more complicated cases (Höglund 2006, p. 37 & p. 40).

The theoretical critical load derived using the energy method introduced above is dependent on support conditions, the load case and the effects of bracing. That means the obtained solution is only relevant for the exact case it was derived for and the method must be repeated if any of the input conditions are changed (Höglund 2006, pp. 58-59).

## 2.4 Elastic critical moment, $M_{cr}$

When finding the lateral-torsional buckling resistance of a beam, a certain maximum theoretical moment is needed which applies for the beam if it was ideal (see Section 2.2.2). That moment is the elastic critical moment, $M_{cr}$. It depends on number of factors, for example the length of the beam, the moment diagram, the support conditions, the stiffness of the beam about the minor axis and the torsional stiffness.

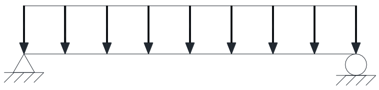
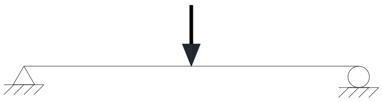
The elastic critical moment is used to find the non-dimensional slenderness of a beam in the process of designing it according to Eurocode 3. However, there is nothing stated about how to determine $M_{cr}$ in Eurocode 3. In an earlier version of Eurocode 3, the pre-standard ENV 1993-1-1:1992, an approximating formula is presented to estimate $M_{cr}$, which gives conservative results. The formula is valid for beams in a major axis bending with a uniform cross section that is symmetric about the minor axis (ECCS 2006, p. 229). Beams in reality are not ideal. That's why a reduction factor must be used to find the design capacity (see section 2.5.1). The formula mentioned above is often called the *3-factor formula* and is expressed as follows:

$$M_{cr} = C_1 \frac{\pi^2 E I_z}{(k_z L)^2} \left\{ \sqrt{\left(\frac{k_z}{k_w}\right)^2 \frac{I_w}{I_z} + \frac{(k_z L)^2 G I_t}{\pi^2 E I_z} + \left(C_2 z_g - C_3 z_j\right)^2} - \left(C_2 z_g - C_3 z_j\right) \right\} \quad (2.26)$$

| | | | |
|---|---|---|---|
| Where | $C_1$ | = | Factor depending on the moment diagram and the end restraints |
| | $C_2$ | = | Factor depending on the moment diagram and the end restraints, related to the vertical position of loading |
| | $C_3$ | = | Factor depending on the moment diagram and the end restraints, related to the monosymmetry of the beam |
| | $E$ | = | Young's modulus of elasticity |
| | $G$ | = | Shear modulus |
| | $I_t$ | = | Torsional constant |
| | $I_w$ | = | Warping constant |
| | $I_z$ | = | Second moment of area about the minor axis |
| | $L$ | = | Length of the beam between points which have lateral restraints |
| | $k_w$ | = | Effective length factor which refers to end warping |
| | $k_z$ | = | Effective length factor which refers to end rotation in plan |
| | $z_g$ | = | Coordinate of the point of load application w.r.t. the shear centre in the z-direction |
| | $z_j$ | = | Monosymmetry parameter |

Values of the factors $C_1$, $C_2$ and $C_3$, referred to as the C-factors, for the two load cases studied in the present theses, given by ECCS (2006) are presented in Table 2.1. Different values for the C-factors can be found in other literature, such as in Access Steel (2010) and Access Steel (2006), where only $\mathrm{k_z} = \mathrm{k_w} = 1$ is considered and in the pre-standard ENV 1993-1-1:1992, where various load cases are considered but some values are overestimated as shown in Mohri et al. (2003).

*Table 2.1* *Values of the factors $C_1$, $C_2$ and $C_3$, corresponding to values of the effective length factors $k_z = k_w$, for the cases of a simply supported beam subjected to a uniformly distributed load and a concentrated mid-span load (adapted from ECCS 2006, Table 64).*

| Load case | $\mathrm{k_z} = \mathrm{k_w}$ | $C_1$ | $C_2$ | $C_3$ |
|---|---|---|---|---|
|  | 1.0 | 1.12 | 0.45 | 0.525 |
| | 0.5 | 0.97 | 0.36 | 0.478 |
|  | 1.0 | 1.35 | 0.59 | 0.411 |
| | 0.5 | 1.05 | 0.48 | 0.338 |

The shear modulus is calculated as follows:

$$G = \frac{E}{2(1+\nu)} \tag{2.27}$$

Where $\nu$ = Poisson's ratio

The parameter $z_g$ describes the vertical position of the PLA. In lateral-torsional buckling, the PLA has a significant influence. If the load acts on the compression flange, i.e. above the SC, the parameter $z_g$ is positive and $M_{cr}$ lower than for $z_g = 0$, so the load has a destabilising effect. If the load acts below the SC, like on the tension flange, the parameter $z_g$ is negative and $M_{cr}$ higher than for $z_g = 0$, so the load has a stabilising effect.

The monosymmetry parameter $z_j$ is defined in Section 2.4.2.1.

In the following sections, simplified versions of the 3-factor formula will be presented for specific cases.

### 2.4.1 I-beams with doubly symmetric cross section

For a beam with a doubly symmetric cross section and fork supports, loaded with equal moments at each end:

$$M_{cr} = \frac{\pi^2 EI_z}{L^2}\sqrt{\frac{I_w}{I_z} + \frac{L^2 GI_t}{\pi^2 EI_z}} \tag{2.28}$$

For a beam with a doubly symmetric cross section and fork supports, loaded at the shear centre:

$$M_{cr} = C_1 \frac{\pi^2 EI_z}{L^2}\sqrt{\frac{I_w}{I_z} + \frac{L^2 GI_t}{\pi^2 EI_z}} \tag{2.29}$$

For a beam with a doubly symmetric cross section and fork supports, loaded along the minor axis above or below the shear centre:

$$M_{cr} = C_1 \frac{\pi^2 EI_z}{L^2}\left\{\sqrt{\frac{I_w}{I_z} + \frac{L^2 GI_t}{\pi^2 EI_z} + \left(C_2 z_g\right)^2} - C_2 z_g\right\} \tag{2.30}$$

### 2.4.2 I-beams with monosymmetric cross section

For a beam with a monosymmetric I-section and fork supports, loaded at the shear centre:

$$M_{cr} = C_1 \frac{\pi^2 EI_z}{L^2}\left\{\sqrt{\frac{I_w}{I_z} + \frac{L^2 GI_t}{\pi^2 EI_z} + \left(C_3 z_j\right)^2} + C_3 z_j\right\} \tag{2.31}$$

For a beam with a monosymmetric I-section and fork supports, loaded along the minor axis above or below the shear centre:

$$M_{cr} = C_1 \frac{\pi^2 EI_z}{L^2}\left\{\sqrt{\frac{I_w}{I_z} + \frac{L^2 GI_t}{\pi^2 EI_z} + \left(C_2 z_g - C_3 z_j\right)^2} - \left(C_2 z_g - C_3 z_j\right)\right\} \tag{2.32}$$

#### 2.4.2.1 Monosymmetry parameters

In the 3-factor formula there is a term $z_j$ that describes the degree of symmetry of the cross section about the y-axis. The term $z_j$ is only nonzero if the cross section is non-symmetric about the y-axis, i.e. monosymmetric, and it is defined as follows:

$$z_j = z_s - 0.5 \int_A (y^2 + z^2) \frac{z}{I_y} dA \tag{2.33}$$

Where $z_s$ = Coordinate of the shear centre w.r.t. the centre of gravity in the z-direction

$A$ = Area of the cross section

$I_y$ = Second moment of area about the strong axis

Equation (2.33) is not particularly suitable for hand calculations. Generally the effect of fillets in the cross section is not taken into account. ECCS (2006) specifies that the following approximation can be used in design (ECCS 2006, p. 231):

$$z_j = \begin{cases} 0.8\psi_f \dfrac{h_s}{2} & \text{for} \quad \psi_f \geq 0 \\ \psi_f \dfrac{h_s}{2} & \text{for} \quad \psi_f < 0 \end{cases} \tag{2.34}$$

Where $h_s$ = Distance between the shear centre of the top flange and the shear centre of the bottom flange of a beam

$\psi_f$ = Differential ratio of the second moments of area about the minor axis for the beam flanges, see equation (2.17)

In some literature the degree of monosymmetry is also described with a parameter that is one of the Wagner's coefficients, which is defined as (Mohri et al. 2003):

$$\beta_z = \frac{1}{2I_y} \int_A z(y^2 + z^2) dA - z_s \tag{2.35}$$

Where $\beta_z$ = The Wagner's coefficient

The Wagner's coefficients can also be found in other literature defined as two times the one defined in equation (2.35) here above (Kitipornchai et al. 1986 & Ziemian 2010), which can result in confusion. It can be seen from comparison of equations (2.33) and (2.35), that the relation between the term $z_j$ and the Wagner's coefficient, $\beta_z$, is (NCS 2009):

$$z_j = -\beta_z \tag{2.36}$$

Various approximating expressions exist to estimate the Wagner's coefficient $\beta_z$ more efficiently. A common expression, adopted in order to match the definition from equation (2.35), is as follows (Kitipornchai et al. 1986):

$$\beta_z = 0.9 h_s \left(\frac{I_{f1}}{I_z} - 0.5\right)\left[1 - \left(\frac{I_z}{I_y}\right)^2\right] \tag{2.37}$$

Where $I_{f1}$ = Second moment of area for the top flange about the minor axis

The accurate expression for the Wagner's coefficient $\beta_z$ formulated for I-sections without fillets, adopted in order to match the definition from equation (2.35), is (Ziemian 2010):

$$\begin{aligned}\beta_z = \frac{1}{2I_y}\Bigg[&\left[\frac{b_{f2}^3}{12} D_1 t_{f2} + \frac{b_{f2}^3}{24} t_{f2}^2 + b_{f2} t_{f2} {D_1}^3 + 1.5 b_{f2} {D_1}^2 t_{f2}^2 \right.\\ &\left. + b_{f2} D_1 t_{f2}^3 + \frac{b_{f2}}{4} t_{f2}^4\right] \\ &- \left[\frac{b_{f1}^3}{12} D_2 t_{f1} + \frac{b_{f1}^3}{24} t_{f1}^2 + b_{f1} t_{f1} {D_2}^3 + 1.5 b_{f1} {D_2}^2 t_{f1}^2 \right.\\ &\left. + b_{f1} D_2 t_{f1}^3 + \frac{b_{f1}}{4} t_{f1}^4\right] \\ &+ \left[\frac{{D_1}^4}{4} t_w + \frac{t_w^3}{24} {D_1}^2 - \frac{{D_2}^4}{4} t_w - \frac{t_w^3}{24} {D_2}^2\right]\Bigg] - z_s\end{aligned} \tag{2.38}$$

Where $D_1$ = Value to determine the Wagner's coefficient $\beta_z$

$D_2$ = Value to determine the Wagner's coefficient $\beta_z$

$b_{f1}$ = Width of the top flange

$t_{f1}$ = Thickness of the top flange

$b_{f2}$ = Width of the bottom flange

$t_{f2}$ = Thickness of the bottom flange

$t_w$ = Thickness of the web

The terms $D_1$ and $D_2$ are calculated as:

$$D_1 = z_G - t_{f2} \tag{2.39}$$

$$D_2 = h_w + t_{f2} - z_G \tag{2.40}$$

Where $z_G$ = Distance from the centre of gravity to the bottom of the cross section

### 2.4.3 Beams with channel cross section

As previously stated, the 3-factor formula, see equation (2.26), is only valid for beams with a cross section that is symmetric about the minor axis. Since beams with a channel section are non-symmetric about the minor axis, the 3-factor formula is not valid for such beams to calculate the elastic critical moment $M_{cr}$.

Various literatures such as ECCS (2006), Access Steel (2010), Access Steel (2006) and ENV 1993-1-1:1992 provide an analytical expression to calculate $M_{cr}$, for cross sections symmetric about the minor axis, i.e. the 3-factor formula. In the said literatures, other types of cross sections are not mentioned. In recent papers, no expression valid for beams with a channel cross section could be found either.

Some commercial structural engineering software, which calculate $M_{cr}$ analytically, such as Colbeam (see Section 3.2.1), use the 3-factor formula anyway for beams with a channel section, as found out by testing.

An additional parameter might be relevant when calculating $M_{cr}$ for beams with a channel section. This parameter describes the degree of symmetry of the cross section about the z-axis and is defined as follows (Mohri et al. 2003):

$$\beta_y = \frac{1}{2I_z}\int_A y(y^2+z^2)dA - y_s \tag{2.41}$$

| | | | |
|---|---|---|---|
| Where | $y_s$ | = | Coordinate of the shear centre w.r.t. the centre of gravity in the y-direction |
| | $I_z$ | = | Second moment of area about the minor axis |
| | $A$ | = | Area of the cross section |
| | $\beta_y$ | = | Wagner's coefficient describing the degree of symmetry about the z-axis |

The parameter $\beta_y$ is one of the Wagner's coefficients and it is analogous to the parameter $\beta_z$, as can be seen from comparison of equations (2.35) and (2.41). However, to the authors' knowledge, no successful attempts have been made to involve $\beta_y$ in the calculation of $M_{cr}$ for beams with a channel section. Therefore its mentioning here in this section is merely informative and might not even be relevant, until further studies have been made.

## 2.5 Design of beams in bending for LT-buckling

Theoretical formulas in capacity calculations are most commonly based on the assumption that beams are ideal, i.e. geometrically perfect and made of linearly elastic material. In reality that is not the case and real beams behave differently than ideal beams, as discussed in Section 2.2.2. Therefore methods including reduction factors are used in order to evaluate a design capacity.

In the following sections an overview will be given on the methods Eurocode 3 offers to design beams in bending with regard to lateral-torsional buckling and also a newly proposed method intended for eccentrically loaded beams with a channel cross section.

### 2.5.1 Design methods in Eurocode 3

As presented in Section 6.3.2.1 in Eurocode 3, for a beam subjected to a major axis bending, lateral-torsional buckling should be verified in the following way:

$$\frac{M_{Ed}}{M_{b,Rd}} \leq 1 \qquad (2.42)$$

| | | | |
|---|---|---|---|
| Where | $M_{Ed}$ | = | Design bending moment |
| | $M_{b,Rd}$ | = | Buckling resistance moment |

The buckling resistance moment is determined as follows:

$$M_{b,Rd} = \chi_{LT} W_y \frac{f_y}{\gamma_{M1}} \qquad (2.43)$$

| | | | |
|---|---|---|---|
| Where | $\chi_{LT}$ | = | Reduction factor for lateral-torsional buckling |
| | $W_y$ | = | Appropriate section modulus, depending on the cross section class |
| | $f_y$ | = | Yield strength of the steel |
| | $\gamma_{M1}$ | = | Partial factor for the resistance of the member to instability assessed by member checks |

For the partial factor $\gamma_{M1}$ Eurocode 3 recommends the value $\gamma_{M1} = 1.0$ for buildings, but $\gamma_{M1}$ may be defined in the National Annex.

#### 2.5.1.1 General case

A general case for calculating the reduction factor $\chi_{LT}$ is given in Section 6.3.2.2 in Eurocode 3 and will be presented here below. If the beam does not have a section that is rolled or an equivalent welded section, the following method is used. Here the reduction factor is:

$$\chi_{LT} = \frac{1}{\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \bar{\lambda}_{LT}^2}} \leq 1 \tag{2.44}$$

| | | | |
|---|---|---|---|
| Where | $\chi_{LT}$ | = | Reduction factor for lateral-torsional buckling |
| | $\Phi_{LT}$ | = | Value to determine the reduction factor $\chi_{LT}$ |
| | $\bar{\lambda}_{LT}$ | = | Non-dimensional slenderness for lateral-torsional buckling |

The non-dimensional slenderness should be taken as:

$$\bar{\lambda}_{LT} = \sqrt{\frac{W_y f_y}{M_{cr}}} \tag{2.45}$$

| | | | |
|---|---|---|---|
| Where | $M_{cr}$ | = | Elastic critical moment for lateral-torsional buckling |
| | $W_y$ | = | Appropriate section modulus, depending on the cross section class |
| | $f_y$ | = | Yield strength of the steel |

And the term $\Phi_{LT}$ as:

$$\Phi_{LT} = 0.5[1 + \alpha_{LT}(\bar{\lambda}_{LT} - 0.2) + \bar{\lambda}_{LT}^2] \tag{2.46}$$

| | | | |
|---|---|---|---|
| Where | $\alpha_{LT}$ | = | Imperfection factor, see Table 2.2 |

The recommended values for the imperfection factor $\alpha_{LT}$ are presented in Table 2.2 for different lateral-torsional buckling curves, but may be defined in the National Annex, and the recommended buckling curves in Table 2.3 for different cross sections.

*Table 2.2 Recommended values for the imperfection factor $\alpha_{LT}$ for lateral-torsional buckling curves used in equation (2.44) (Eurocode 3, Table 6.3).*

| Buckling curve | a | b | c | d |
|---|---|---|---|---|
| Imperfection factor $\alpha_{LT}$ | 0.21 | 0.34 | 0.49 | 0.76 |

*Table 2.3 Recommended lateral-torsional buckling curves for cross sections used with equation (2.44) (Eurocode 3, Table 6.4). h represents the height of the beam and $b_f$ represents the width of the beam.*

| Cross section | Limits | Buckling curve |
|---|---|---|
| Rolled I-sections | $h/b_f \leq 2$<br>$h/b_f > 2$ | a<br>b |
| Welded I-sections | $h/b_f \leq 2$<br>$h/b_f > 2$ | c<br>d |
| Other cross sections | - | d |

#### 2.5.1.2 Rolled sections or equivalent welded sections

A case for rolled sections or equivalent welded sections for calculating the reduction factor $\chi_{LT}$ is given in Section 6.3.2.3 in Eurocode 3 and will be presented here below. For a beam with a rolled section or an equivalent welded section, the reduction factor is:

$$\chi_{LT} = \frac{1}{\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \beta\bar{\lambda}_{LT}^2}} \quad \text{but} \quad \begin{cases} \chi_{LT} \leq 1 \\ \chi_{LT} \leq \sqrt{\dfrac{1}{\bar{\lambda}_{LT}^2}} \end{cases} \tag{2.47}$$

Where $\Phi_{LT}$ = Value to determine the reduction factor $\chi_{LT}$

$\chi_{LT}$ = Reduction factor for lateral-torsional buckling

$\beta$ = Correction factor for the lateral-torsional buckling curves for rolled sections

$\bar{\lambda}_{LT}$ = Non-dimensional slenderness for lateral-torsional buckling

For the correction factor $\beta$ Eurocode 3 recommends the value $\beta = 0.75$ as a minimum value, but $\beta$ may be defined in the National Annex.

The term $\Phi_{LT}$ should be taken as:

$$\Phi_{LT} = 0.5[1 + \alpha_{LT}(\bar{\lambda}_{LT} - \bar{\lambda}_{LT,0}) + \beta\bar{\lambda}_{LT}^2] \tag{2.48}$$

Where $\bar{\lambda}_{LT,0}$ = Plateau length of the lateral-torsional buckling curves for rolled sections

$\alpha_{LT}$ = Imperfection factor, see Table 2.2

For the plateau length $\bar{\lambda}_{LT,0}$ Eurocode 3 recommends the value $\bar{\lambda}_{LT,0} = 0.4$ as a maximum value, but $\bar{\lambda}_{LT,0}$ may be defined in the National Annex.

The recommended buckling curves for different cross sections are presented in Table 2.4. The imperfection factor $\alpha_{LT}$ is determined in the same way as for the general case, i.e. with Table 2.2, but may be defined in the National Annex.

*Table 2.4 Recommended lateral-torsional buckling curves for cross sections used with equation (2.47) (Eurocode 3, Table 6.5). h represents the height of the beam and $b_f$ represents the width of the beam.*

| Cross section | Limits | Buckling curve |
|---|---|---|
| Rolled I-sections | $h/b_f \leq 2$<br>$h/b_f > 2$ | b<br>c |
| Welded I-sections | $h/b_f \leq 2$<br>$h/b_f > 2$ | c<br>d |

The lateral-torsional buckling needs not to be checked if the following applies:

$$\bar{\lambda}_{LT} \leq \bar{\lambda}_{LT,0} \tag{2.49}$$

Or, expressed in another way:

$$\frac{M_{Ed}}{M_{cr}} \leq \bar{\lambda}_{LT,0}^2 \tag{2.50}$$

Where $M_{Ed}$ = Design bending moment

#### 2.5.1.3 Simplified method

A simplified method is presented in Section 6.3.2.4 in Eurocode 3, for beams with discrete lateral restraints to the compression flange. In this method the focus is on the compression flange, and if it plus a part of the compression zone in the web can withstand the resulting moment without buckling laterally. The buckling resistance in this method is taken as:

$$M_{b,Rd} = k_{f\ell}\chi M_{c,Rd} \quad \text{but} \quad M_{b,Rd} \leq M_{c,Rd} \tag{2.51}$$

Where $\chi$ = Reduction factor of the equivalent compression flange determined with $\bar{\lambda}_f$

$k_{f\ell}$ = Modification factor accounting for the conservatism of the equivalent compression flange method

| | | | |
|---|---|---|---|
| | $M_{b,Rd}$ | = | Design buckling resistance moment |
| | $M_{c,Rd}$ | = | Design resistance for bending about one principal axis of a cross section |

The design resistance is:

$$M_{c,Rd} = W_y \frac{f_y}{\gamma_{M1}} \tag{2.52}$$

| | | | |
|---|---|---|---|
| Where | $W_y$ | = | Appropriate section modulus corresponding to the compression flange |
| | $\gamma_{M1}$ | = | Partial factor for the resistance of the member to instability assessed by member checks |

For the modification factor $k_{f\ell}$ Eurocode 3 recommends the value $k_{f\ell} = 1.1$, but $k_{f\ell}$ may be defined in the National Annex.

The reduction factor $\chi$ is calculated with the same method as in Section 2.5.1.1, but determined with the relative slenderness $\bar{\lambda}_f$:

$$\bar{\lambda}_f = \frac{k_c L}{i_{f,z}\bar{\lambda}_1} \tag{2.53}$$

| | | | |
|---|---|---|---|
| Where | $i_{f,z}$ | = | Radius of gyration of the equivalent compression flange composed of the compression flange plus $1/3$ of the compressed part of the web area, about the minor axis of the section |
| | $k_c$ | = | Slenderness correction factor for the moment distribution between restraints |
| | $L$ | = | Length between lateral restraints of the compression flange |
| | $\bar{\lambda}_1$ | = | Slenderness value to determine the relative slenderness |
| | $\bar{\lambda}_f$ | = | Equivalent compression flange slenderness |

The term $\lambda_1$ is calculated as:

$$\bar{\lambda}_1 = \pi \sqrt{\frac{E}{f_y}} \tag{2.54}$$

| | | | |
|---|---|---|---|
| Where | $E$ | = | Young's modulus of elasticity |

The radius of gyration for sections in classes 1, 2 and 3 is calculated as:

$$i_{f,z} = \sqrt{\frac{I_f}{A_f + \frac{1}{3} A_{w,c}}} \tag{2.55}$$

| | | | |
|---|---|---|---|
| Where | $A_f$ | = | Area of the flange in compression |
| | $A_{w,c}$ | = | Area of the part of the web which is in compression |
| | $I_f$ | = | Second moment of area of the flange in compression about the minor axis of the beam |

The sections of the beams used in this thesis are in class 1. The radius of gyration for class 4 sections is thus not of interest, where the effective sectional parameters must be used in equation (2.55).

If the following inequation applies for the relative slenderness $\bar{\lambda}_f$, the compression flange is not susceptible to LT-buckling:

$$\bar{\lambda}_f \leq \bar{\lambda}_{c0} \frac{M_{c,Rd}}{M_{y,Ed}} \tag{2.56}$$

| | | | |
|---|---|---|---|
| Where | $M_{c,Rd}$ | = | See equation (2.52) |
| | $M_{y,Ed}$ | = | See equation (2.60) |
| | $\bar{\lambda}_{c0}$ | = | Slenderness limit of the equivalent compression flange defined under equation (2.53) |

The slenderness limit $\bar{\lambda}_{c0}$ may be defined in the National Annex, but for $\bar{\lambda}_{c0}$ Eurocode 3 recommends a limit value calculated as follows:

$$\bar{\lambda}_{c0} = \bar{\lambda}_{LT,0} + 0.1 \tag{2.57}$$

| | | | |
|---|---|---|---|
| Where | $\bar{\lambda}_{LT,0}$ | = | Plateau length of the lateral-torsional buckling curves for rolled sections |

#### 2.5.1.4 General method

For beams with a complicated cross section or support conditions, which other methods Eurocode 3 offers to design beams in bending with regard to lateral-torsional buckling do not apply for, a general method presented in Section 6.3.4 in Eurocode 3 can be used. The general method can be used to check for stability against flexural buckling due to minor axis bending and lateral-torsional buckling.

The resistance of out of plane buckling is sufficient according to the general method if the following is fulfilled:

$$\frac{\chi_{op}\alpha_{ult,k}}{\gamma_{M1}} \geq 1 \qquad (2.58)$$

Where $\alpha_{ult,k}$ = Minimum load amplifier of the design loads to reach the characteristic resistance of the most critical cross section

$\chi_{op}$ = Reduction factor for the non-dimensional slenderness, $\bar{\lambda}_{op}$

The non-dimensional slenderness, $\bar{\lambda}_{op}$ is determined by:

$$\bar{\lambda}_{op} = \sqrt{\frac{\alpha_{ult,k}}{\alpha_{cr,op}}} \qquad (2.59)$$

Where $\alpha_{cr,op}$ = Minimum amplifier for the in plane design loads to reach the elastic critical resistance with regard to flexural buckling due to minor axis bending and lateral-torsional buckling

$\bar{\lambda}_{op}$ = Global non-dimensional slenderness of a structural component for out-of-plane buckling

To explain the $\alpha$-terms in equation (2.59) better it is referred to Eurocode 3 where it is described in this way, adapted to the usage of terms in this thesis:

$\alpha_{ult,k}$ is the minimum load amplifier of the design loads to reach the characteristic resistance of the most critical cross section of the structural component considering its in plane behaviour without taking flexural buckling, due to minor axis bending, or lateral-torsional buckling into account, however accounting for all effects due to in plane geometrical deformation and imperfections, global and local, where relevant.

$\alpha_{cr,op}$ is the minimum amplifier for the in plane design loads to reach the elastic critical resistance of the structural component with regards to flexural buckling, due to minor axis bending, or lateral-torsional buckling without accounting for flexural buckling due to minor axis bending.

FEM may be used to determine $\alpha_{ult,k}$ and $\alpha_{cr,op}$. For lateral-torsional buckling, the $\chi_{op}$ may be taken as the minimum value of $\chi_{LT}$ in Section 2.5.1.1 using the non-dimensional slenderness $\bar{\lambda}_{op}$ instead of $\bar{\lambda}_{LT}$.

In the present thesis axial forces are not considered and will be neglected from the final formula that should be fulfilled:

$$\chi_{op} \geq \frac{M_{y,Ed}}{M_{y,Rk}/\gamma_{M1}} \tag{2.60}$$

Where $M_{y,Ed}$ = Design bending moment about the y-axis

$M_{y,Rk}$ = Characteristic bending resistance about the y-axis

### 2.5.2 Beams with channel sections

When designing a beam with a channel section, Eurocode 3 does not suggest any specific method. It is stated though, that the general case can be used with buckling curve d for “other cross sections”. When centrically loaded, i.e. loaded at the shear centre, beams with a channel section can be designed for lateral-torsional buckling using methods for doubly symmetric beams (Snijder et al. 2008).

However, Snijder et al. (2008) propose a design rule for eccentrically loaded beams with a channel section, which is presented in Section 6 in their paper, by adding a torsional term to the relative slenderness of the beam. To verify that the beam has enough lateral-torsional buckling capacity, the following should be fulfilled:

$$\frac{M_{Ed}}{\chi_{LT} \cdot M_{pl}} \leq 1 \tag{2.61}$$

Where $M_{Ed}$ = Design bending moment

$M_{pl}$ = Plastic moment capacity of the cross section

$\chi_{LT}$ = Reduction factor for lateral-torsional buckling

The reduction factor is found using the same method as in the general case in Eurocode 3 (see Section 2.5.1.1), except here $\bar{\lambda}_{MT}$ is used instead of $\bar{\lambda}_{LT}$. The $\bar{\lambda}_{MT}$ is the relative slenderness for buckling with the influence from torsion:

$$\bar{\lambda}_{MT} = \bar{\lambda}_M + \bar{\lambda}_T \tag{2.62}$$

Where

$$\bar{\lambda}_M = \sqrt{\frac{M_{pl}}{M_{cr}}} \tag{2.63}$$

And

$$\bar{\lambda}_T = \begin{cases} 1.0 - \bar{\lambda}_M & \text{if} \quad 0.5 \leq \bar{\lambda}_M \leq 0.8 \\ 0.43 - 0.29\bar{\lambda}_M & \text{if} \quad 0.8 \leq \bar{\lambda}_M \leq 1.5 \\ 0 & \text{if} \quad \bar{\lambda}_M \geq 1.5 \end{cases} \tag{2.64}$$

In fact the $\bar{\lambda}_M$ is the same as $\bar{\lambda}_{LT}$ in Eurocode 3, see Section 2.5.1.1. The additional torsional term, $\bar{\lambda}_T$, results in the following reduction factor:

$$\chi_{LT} = \frac{1}{\Phi_{LT} + \sqrt{\Phi_{LT}^2 - \bar{\lambda}_{MT}^2}} \leq 1 \tag{2.65}$$

| | | | |
|---|---|---|---|
| Where | $\Phi_{LT}$ | = | Value to determine the reduction factor $\chi_{LT}$, see equation (2.46) |
| | $\alpha_{LT}$ | = | 0.21 (buckling curve a) |

The factor $\Phi_{LT}$ is to be calculated as in the general case (see Section 2.5.1.1), i.e. with equation (2.46).

# 3 Software Introduction

In this chapter the following five commercial structural engineering software tools will be introduced:

- ADINA-AUI 900 Nodes v9.0
- Colbeam EC3 v1.1.8
- LTBeam v1.0.11
- SAP2000 v15
- STAAD.Pro V8i (SELECTseries 3) v20.07.08.20

The software will hereinafter be referred to as ADINA, Colbeam, LTBeam, SAP2000 and STAAD.Pro respectively. Information about their background will be given, their methods for finding the elastic critical moment $M_{cr}$ described and their possibilities and limitations in that area discussed.

## 3.1 ADINA

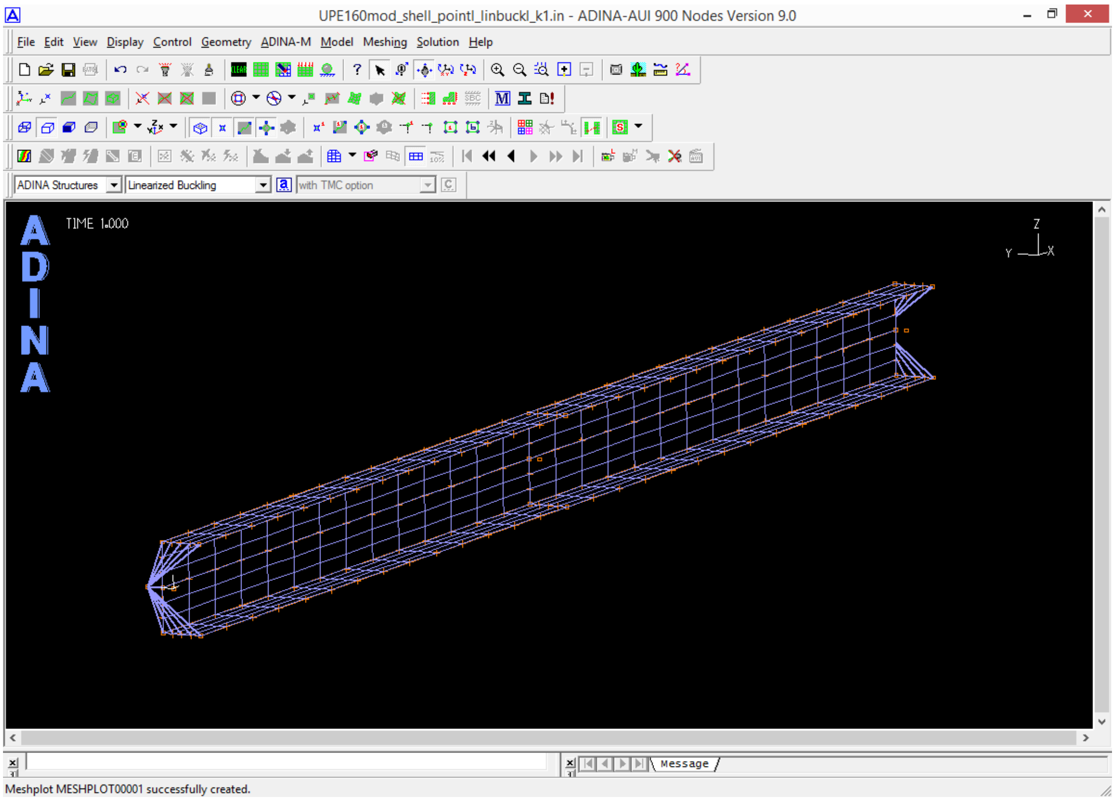



*Figure 3.1 ADINA user interface showing a model of a channel beam*

The development of ADINA started in 1974, the same year as the foundation of the company was established. The name was changed to ADINA R&D, Inc., which is the present name of the company (ADINA R&D, n.d.).

ADINA is a FEM based software which has many possibilities with different complexity levels to approach a problem.

The software gives possibilities to model both simple and complex structures. By choosing the appropriate type, number, size and shape of elements, number of integration points, integration method, nodes per elements etc., the user is able to have a good control over the model, both how it is modelled and analysed.

The main disadvantages of this software are the graphics and the user interface, which can be seen in Figure 3.1. To be able to understand or check the model, the user has to look into command or output files, which are not user friendly, or search the tabs in the software. The user interface does not have a certain modelling path to follow (like for example ABAQUS or STAAD.Pro) so one has to go back and forth in ones way to make a complete model.

### 3.1.1 Elastic critical moment

ADINA can evaluate the critical load in two ways; using a linearized buckling analysis or a non-linear incremental collapse analysis with the load-displacement-control (LDC) method. When doing a FEM analysis for a structure, often a linearized buckling analysis is first performed and the resulting buckling mode shapes used as geometric imperfection in a collapse analysis with the LDC method (ADINA 2013, p. 848).

#### 3.1.1.1 Linearized buckling analysis

Linearized buckling analysis in ADINA is a type of a FEM analysis where the solution is obtained by solving a generalized eigenvalue problem. Two different formulations can be chosen; the classical formulation and the secant formulation. The eigenvalue problem for the classical formulation is (ADINA 2013, p. 845):

$$K^{t_1}\phi_i = \gamma_i\left(K^{t_1} - K_{NL}^{t_1}\right)\phi_i, \quad \gamma_i > 0 \tag{3.1}$$

And for the secant formulation:

$$K^{t_1}\phi_i = \gamma_i K^{t_0}\phi_i, \quad \gamma_i > 0 \tag{3.2}$$

| | | | |
|---|---|---|---|
| Where | $K^{t_1}$ | = | Stiffness matrix of the structural system at time $t_1$ |
| | $K^{t_0}$ | = | Stiffness matrix of the structural system at time $t_0$ |
| | $K_{NL}^{t_1}$ | = | Geometrically nonlinear part of $K^{t_1}$ |
| | $\phi_i$ | = | Buckling mode shape for mode $i$ |
| | $\gamma_i$ | = | Eigenvalue for mode $i$ |

The critical buckling load factor is related to the eigenvalue by:

$$\lambda_i = \frac{1}{1-\gamma_i} \tag{3.3}$$

Where $\lambda_i$ = Critical buckling load factor for mode $i$

The classical buckling formulation is better than the secant formulation because of its more accurate computation (ADINA 2013, pp. 846). The eigenvalues are obtained and arranged in the following order, with $N$ as the requested number of eigenvalues:

$$\gamma_1 \leq \gamma_2 \leq \cdots \leq \gamma_M \leq 1 \leq \gamma_{M+1} \leq \cdots \leq \gamma_N \tag{3.4}$$

The following then applies for the load factors:

$$1 \leq \lambda_1 \leq \lambda_2 \leq \cdots \leq 1 \leq \lambda_M \tag{3.5}$$

and

$$\lambda_{M+1} \leq \lambda_{M+2} \leq \cdots \leq \lambda_N \leq 0 \tag{3.6}$$

The linearized buckling analysis is an approximation (R. Wang, personal correspondence, March 19, 2014) and should only be used for the simplest cases of bifurcation buckling (Earls 2007). When running a linearized buckling analysis, a load must be specified in order for the software to be able to calculate the buckling load. ADINA returns a value, a critical load factor $\lambda$, which should be multiplied with the applied load to obtain the linearized buckling load. The calculated buckling load varies with different applied load. The results are more accurate as the applied load increases (Earls 2007).

For the first buckling mode the critical buckling load factor, $\lambda_1$, must fulfil the following condition in order to execute the analysis (ADINA 2013, pp. 846):

$$1 < \lambda_1 < 500 \tag{3.7}$$

Since the $\lambda_1$ factor is multiplied to the value of applied load to obtain the critical buckling load for mode 1, this means it is not possible to apply a load higher than or equal to the buckling load.

#### 3.1.1.2 Collapse analysis with the LDC method

The collapse analysis with the load-displacement-control (LDC) method is a nonlinear incremental static analysis that can calculate both pre-collapse and post-collapse response of a model. The analysis utilises the arc-length method so it automatically adjusts the level of externally applied load in each time step. In the analysis an initial user defined displacement is imposed on a specified degree of freedom to evaluate the initial load vector. The analysis continues by applying additional load in each time step until a specified displacement is reached in some node or number of converged solution steps is reached. The user can also choose to make the analysis stop when a critical point on the equilibrium path is reached. In each iteration, the following equilibrium equations are used (ADINA 2013, pp. 876-877):

$$K_{i-1}^{t+\Delta t}\Delta U_i = \left(\lambda_{i-1}^{t+\Delta t} + \Delta\lambda_i\right)R + R_p - F_{i-1}^{t+\Delta t} \tag{3.8}$$

$$U_i^{t+\Delta t} = U_{i-1}^{t+\Delta t} + \Delta U_i \tag{3.9}$$

$$f(\Delta\lambda_i\,, \Delta U_i) = 0 \tag{3.10}$$

| | | | |
|---|---|---|---|
| Where | $K_{i-1}^{t+\Delta t}$ | = | Tangent stiffness matrix at time $t+\Delta t$ and end of iteration number $i-1$ |
| | $U_i^{t+\Delta t}$ | = | Displacement vector at time $t+\Delta t$ and end of iteration number $i$ |
| | $\Delta U_i$ | = | Incremental displacement vector in iteration number $i$ |
| | $R$ | = | Constant reference load vector |
| | $R_p$ | = | Load vector from previous solution run |
| | $\lambda_{i-1}^{t+\Delta t}$ | = | Load scaling factor applied on $R$ at time $t+\Delta t$ and end of iteration number $i-1$ |
| | $\Delta\lambda_i$ | = | Increment in the load scaling factor in iteration number $i$ |
| | $F_{i-1}^{t+\Delta t}$ | = | Nodal force vector at time $t+\Delta t$ and end of iteration number $i-1$ |

Equation (3.10) is used to constrain the length of each time step in the arc-length method. It should be noted that the equilibrium path the solution to an analysis with the LDC method takes, is highly dependent on the magnitude and especially the direction of the initial prescribed displacement. When the LDC method is employed, the load scaling factor $\lambda_i^t$ is used to scale the applied load $R$ instead of the loading being user controlled through a so called time function (ADINA 2013, pp. 878-879).

For reaching a convergence in each time step in the equilibrium iteration process, the following criteria can be used (ADINA 2013, pp. 880):

- Energy
- Energy and force/moment
- Energy and translation/rotation
- Force/moment
- Translation/rotation

The convergence tolerance can be user defined in all cases. For most cases the energy convergence criterion is sufficient if the tolerance is small enough (ADINA 2013, pp. 888).

The collapse analysis with the LDC method can be suitable for the more complicated buckling problems, compared to a linearized buckling analysis, but requires much more computational effort. It cannot be considered ideal for the regular design procedure of a beam.

In order to estimate the critical load, a load-displacement graph can be made, similar to Figure 2.14. If a bifurcation point exists, the corresponding load is the critical load.

### 3.1.2 Calculation of sectional parameters

ADINA uses a set of integration formulas to calculate the sectional parameters corresponding to a beam modelled with beam elements, where integration over the cross section has to be made. The formulas are presented here below, where they have been adapted to match the coordinate system used in this thesis (see Section 2.1.1), since ADINA uses the symbols s and t for the y- and z-directions respectively (ADINA 2013, pp. 68-70):

Cross sectional area:

$$A = \int_A dA \tag{3.11}$$

Second moments of area about the major axis:

$$I_y = \int_A z^2 dA \tag{3.12}$$

Second moment of area about the minor axis:

$$I_z = \int_A y^2 dA \tag{3.13}$$

Product of second moment of area:

$$I_{yz} = \int_A yz dA \tag{3.14}$$

Saint-Venant torsional constant:

$$I_t = \int_A \left( \left( \frac{\partial \psi}{\partial y} - (z - z_s) \right)^2 + \left( \frac{\partial \psi}{\partial z} + (y - y_s) \right)^2 \right) dA \tag{3.15}$$

Warping constant:

$$I_w = \int_A \psi^2 dA \tag{3.16}$$

Wagner effect constants:

$$A_{15} = \int_A ((y - y_s)^2 + (z - z_s)^2) dA \tag{3.17}$$

$$A_{25} = \int_A y((y - y_s)^2 + (z - z_s)^2) dA \tag{3.18}$$

$$A_{35} = \int_A z((y - y_s)^2 + (z - z_s)^2) dA \tag{3.19}$$

$$A_{45} = \int_A \psi((y - y_s)^2 + (z - z_s)^2) dA \tag{3.20}$$

$$A_{55} = \int_A ((y - y_s)^2 + (z - z_s)^2)^2 dA \tag{3.21}$$

To make use of these formulas to calculate the sectional parameters by hand, the warping function $\psi$ needs to be established. For a thin walled cross section the warping function can be established for each cross section part (see Figure 3.2) by evaluating the following line integral (ADINA 2013, p. 87):

$$\psi(y, z) = -\int_0^q \hat{p} dq + p\hat{q} \tag{3.22}$$

The notations used in equation (3.22) are explained in Figure 3.2. The $pq$-coordinate system has an origin at the cross section parts centroid and the $q$-axis lies along the parts longitudinal axis. The $\hat{p}\hat{q}$-coordinate system has an origin at the shear centre of the cross section and the $\hat{q}$-axis lies parallel to the longitudinal axis of the cross section part.

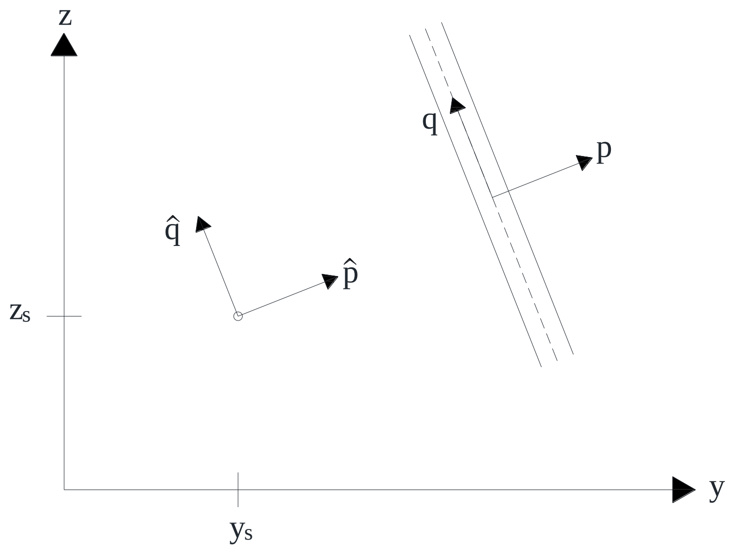



*Figure 3.2 Illustration of the notations used to establish the warping function for a thin walled cross section element (adapted from ADINA 2013, p. 86).*

## 3.2 Colbeam

Colbeam is a member design program from StruProg AB. In the present thesis the Eurocode version will be used and its user interface can be seen in Figure 3.3. The software is also available for the Swedish BSK 07 code as well as the Norwegian NS3472 code (StruProg 2013).

The load can be applied as an axial load, a transverse load in the gravity and/or the horizontal directions, a moment and a distributed load. It is not possible to have more than one span. Instead the user can use spring supports to obtain similar conditions as if it was one of the span in a multi-span beam. Rolled I-beams, beams with a channel sections, rectangular tubes and L-shaped beams in most common sizes are possible to choose. Welded beams are also an option, but then the geometry must be inserted manually. It is possible to manually put in values for sectional properties such as $I_z$, $I_w$ and $I_t$, but then the section will be treated as a doubly symmetric I-section.

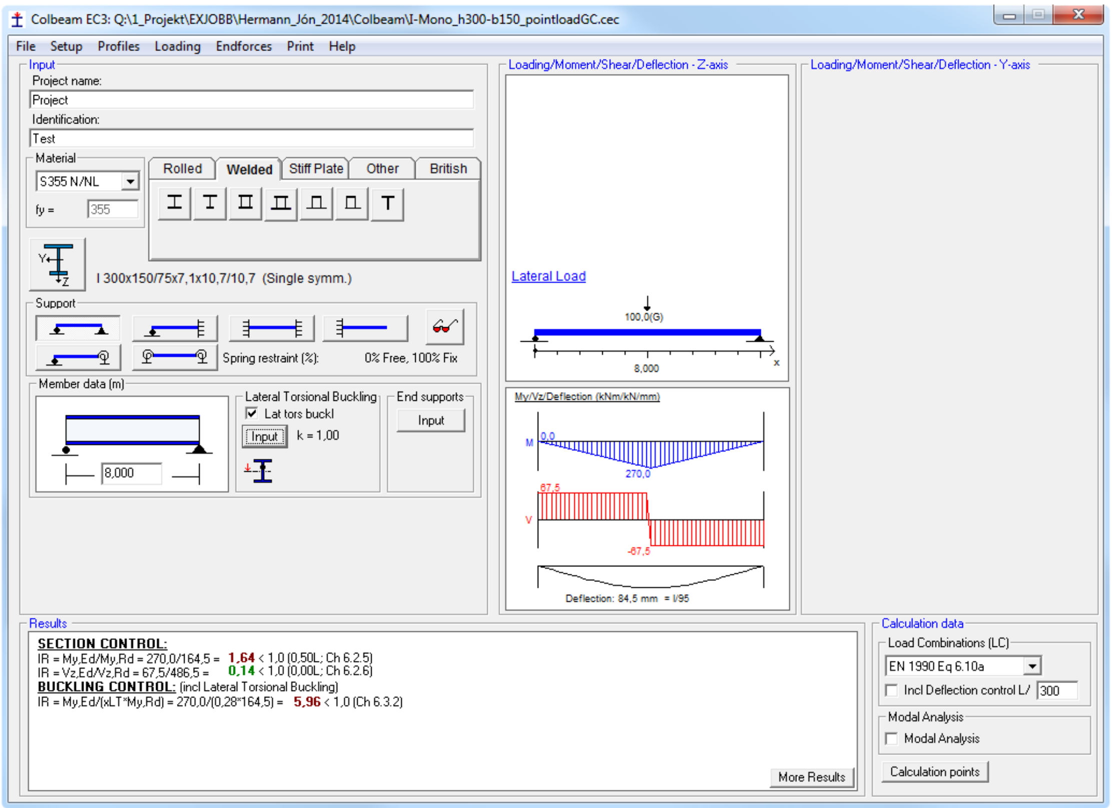
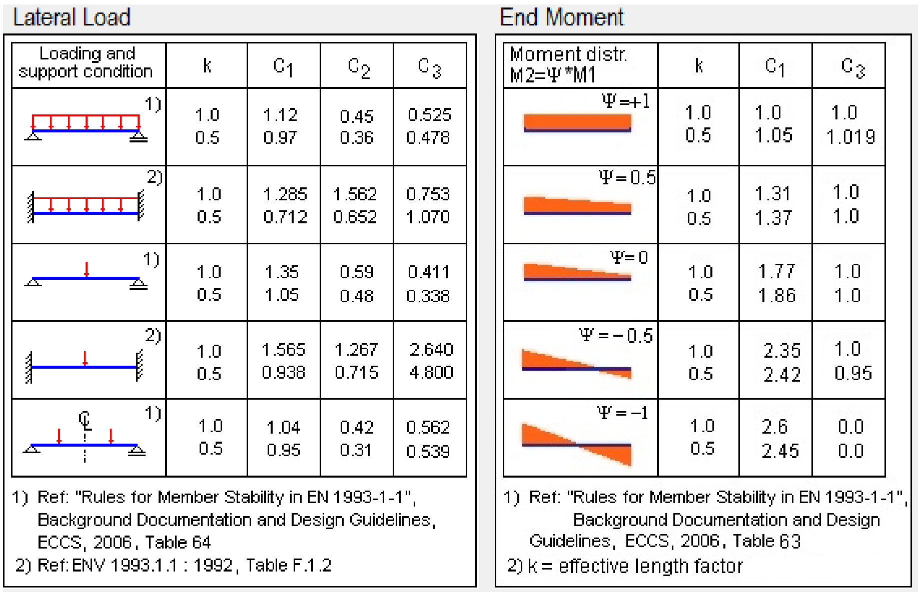



*Figure 3.3 The user interface of Colbeam.*

## 3.2.1 Elastic critical moment



Lateral Load

| Loading and support condition | k | $C_1$ | $C_2$ | $C_3$ |
|---|---|---|---|---|
| 1) | 1.0<br>0.5 | 1.12<br>0.97 | 0.45<br>0.36 | 0.525<br>0.478 |
| 2) | 1.0<br>0.5 | 1.285<br>0.712 | 1.562<br>0.652 | 0.753<br>1.070 |
| 1) | 1.0<br>0.5 | 1.35<br>1.05 | 0.59<br>0.48 | 0.411<br>0.338 |
| 2) | 1.0<br>0.5 | 1.565<br>0.938 | 1.267<br>0.715 | 2.640<br>4.800 |
| 1) | 1.0<br>0.5 | 1.04<br>0.95 | 0.42<br>0.31 | 0.562<br>0.539 |

1) Ref: "Rules for Member Stability in EN 1993-1-1", Background Documentation and Design Guidelines, ECCS, 2006, Table 64

2) Ref: ENV 1993.1.1 : 1992, Table F.1.2

End Moment

| Moment distr. M2=Ψ*M1 | k | $C_1$ | $C_3$ |
|---|---|---|---|
| Ψ=+1 | 1.0<br>0.5 | 1.0<br>1.05 | 1.0<br>1.019 |
| Ψ=0.5 | 1.0<br>0.5 | 1.31<br>1.37 | 1.0<br>1.0 |
| Ψ=0 | 1.0<br>0.5 | 1.77<br>1.86 | 1.0<br>1.0 |
| Ψ=−0.5 | 1.0<br>0.5 | 2.35<br>2.42 | 1.0<br>0.95 |
| Ψ=−1 | 1.0<br>0.5 | 2.6<br>2.45 | 0.0<br>0.0 |

1) Ref: "Rules for Member Stability in EN 1993-1-1", Background Documentation and Design Guidelines, ECCS, 2006, Table 63

2) k = effective length factor

*Figure 3.4 The built-in values for the $C_1$, $C_2$ and $C_3$ factors for specific load cases in Colbeam.*

Colbeam employs analytical methods for calculations and uses the 3-factor formula, see equation (2.26), to calculate the elastic critical moment for lateral-torsional buckling. The 3-factor formula is for beams with uniform cross sections that are symmetric about the minor axis (ECCS 2006, p. 229). In spite of that, as found out by testing, Colbeam uses the 3-factor formula to calculate $M_{cr}$ for beams with channel sections, which do not fulfil that requirement.

Colbeam automatically selects values for the $C_1$, $C_2$ and $C_3$ factors for the load case specified and the built-in load cases can be seen in Figure 3.4. In the case of $k_z = 0.7$, Colbeam obtains the values for the C-factors by linear interpolation between the values for $k_z = 1$ and $k_z = 0.5$.

The built-in values for the C-factors are based on the values from ECCS (2006), Table 63 and Table 64. Not all cases can be found there, but the remaining values are taken from ENV 1993-1-1:1992, Table F1.2. Those remaining values are for beams with fixed ends, both for a point load and a distributed load. An exception from this is not stated in any help or literature the software provides, which is the determination of $C_3$ in case of an end moment loading. When the proportion between the end moments is $\psi = -1/2$ (see Figure 3.5), only the values for the case when $\psi_f \leq 0$ (see equation (2.17)) are used. For the case when $\psi = -1$, the values are taken from ENV 1993-1-1:1992, not ECCS (2006).

Some values for the C-factors used by Colbeam from the pre-standard ENV 1993-1-1:1992 are quite different than values for the same cases from newer sources, such as Access Steel (2010). For example does the $C_1$ value Colbeam uses in the case of a beam with fixed end supports and distributed loading seem to give unreasonable results (Ahnlén & Westlund 2013, pp. 73-74).


*Figure 3.5 Illustration of a beam with end moment loading. The magnitude of the moment applied at one end is $M$ and $\psi \cdot M$ at the other.*

Manual input of the C-factors is also possible. Whether the built-in values or manually inserted values are used, the factor $C_1$ can be approximated by choosing an option of a so called advanced calculation of $C_1$. The advanced option is based on a closed form expression that López et. al (2006) published (StruProg AB 2010). Since the tabulated values for the C-factors have been derived for specific load cases, they should give a more accurate result for those cases, than values from a closed form expression. The formula to evaluate $C_1$ using the closed form expression:

$$C_1 = \frac{\sqrt{\sqrt{k}A_1 + \left[\frac{1-\sqrt{k}}{2}A_2\right]^2} + \frac{1-\sqrt{k}}{2}A_2}{A_1} \tag{3.23}$$

The parameter $k$ is defined as:

$$k = \sqrt{k_1 k_2} \tag{3.24}$$

| | | | |
|---|---|---|---|
| Where | $k_1$ | = | Buckling length factor on the left end seen from the user, representing $k_w$ and $k_z$ |
| | $k_2$ | = | Buckling length factor on the right end seen from the user, representing $k_w$ and $k_z$ |
| | $k$ | = | Buckling length factor dependent on $k_1$ and $k_2$ |

It should be noted that Colbeam assumes $k_w = k_z$. The parameters $A_1$ and $A_2$ are determined as follows:

$$A_1 = \frac{M_{max}^2 + \alpha_1 M_1^2 + \alpha_2 M_2^2 + \alpha_3 M_3^2 + \alpha_4 M_4^2 + \alpha_5 M_5^2}{(1 + \alpha_1 + \alpha_2 + \alpha_3 + \alpha_4 + \alpha_5) M_{max}^2} \tag{3.25}$$

And

$$A_2 = \left| \frac{M_1 + 2M_2 + 3M_3 + 2M_4 + M_5}{9M_{max}} \right| \tag{3.26}$$

| | | | |
|---|---|---|---|
| Where | $M_{max}$ | = | Maximum moment acting on a beam |
| | $M_1$ | = | Moment value acting on a beam at the left end seen from the user |
| | $M_2$ | = | Moment value acting on a beam at the point which is $0.25L$ from the left end seen from the user |
| | $M_3$ | = | Moment value acting on a beam at the centre of the span |
| | $M_4$ | = | Moment value acting on a beam at the point which is $0.25L$ from the right end seen from the user |
| | $M_5$ | = | Moment value acting on a beam at the right end seen from the user |

A graphical explanation of the moment values $M_1$ to $M_5$ can be found in Figure 3.6.

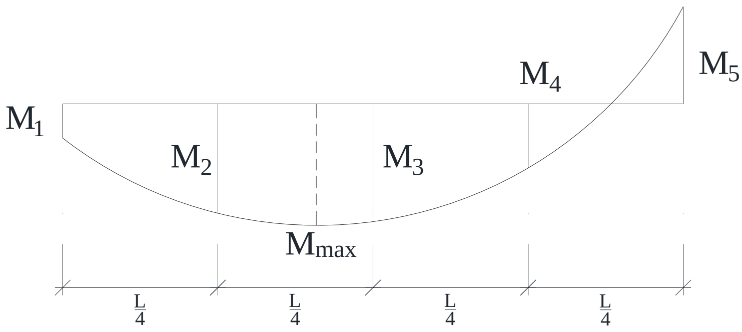



*Figure 3.6* *Illustration of an example of a moment diagram and the values of bending moment needed to calculate the parameters $A_1$ and $A_2$ in equations (3.25) and (3.26).*

The $\alpha$ values are defined as follows:

$$\alpha_1 = 1 - k_2 \tag{3.27}$$

$$\alpha_2 = 5\frac{k_1^3}{k_2^2} \tag{3.28}$$

$$\alpha_3 = 5\left(\frac{1}{k_1} + \frac{1}{k_2}\right) \tag{3.29}$$

$$\alpha_4 = 5\frac{k_2^3}{k_1^2} \tag{3.30}$$

$$\alpha_5 = 1 - k_1 \tag{3.31}$$

For built-in sections the sectional parameters, such as $I_w$ and $I_t$, have built-in values. When manually entering the geometry, they are calculated as in equations (2.19) and (2.9), respectively. The monosymmetry parameter $z_j$ is calculated using the approximate expression from ECCS (2006), see equation (2.34).

As mentioned earlier, Colbeam assumes $k_w = k_z$, see Section 2.1.8.

## 3.3 LTBeam

LTBeam is a software used in the field of the computational structural steel. It was developed by CTICM within a European research project, partly funded by the European Coal and Steel Community (ECSC), and completed in 2002 (CTICM, n.d.).

LTBeam is a software that deals with elastic lateral-torsional buckling of beams subjected to bending about their major axis (CTICM, n.d.). It uses FEM to obtain it´s results.

This software applies to straight beams with one or more spans under simple bending about their major axis. The cross section must be symmetric about the minor axis, but can vary along the x-axis. Each node has four degrees of freedom (DOFs). At the beam ends, the user can choose which one of them to restrain. The four DOFs are:

- Lateral displacement
- Torsion rotation
- Lateral flexural rotation
- Warping

Along the beam the following restraints can be applied:

- Lateral displacement, local and continuous
- Torsional rotation, local and continuous

Supports conditions in the bending plane and externally applied loads are taken into account by the bending moment distribution. Two input methods are available; the simple input mode and the file input mode, shown in Figure 3.7 and Figure 3.8 respectively. As the name indicates the simple input mode is simpler and therefore faster. It is unnecessary to use the file input mode unless the beam to be checked has a variable cross section or if complex loading conditions are present.

### 3.3.1 Simple input mode

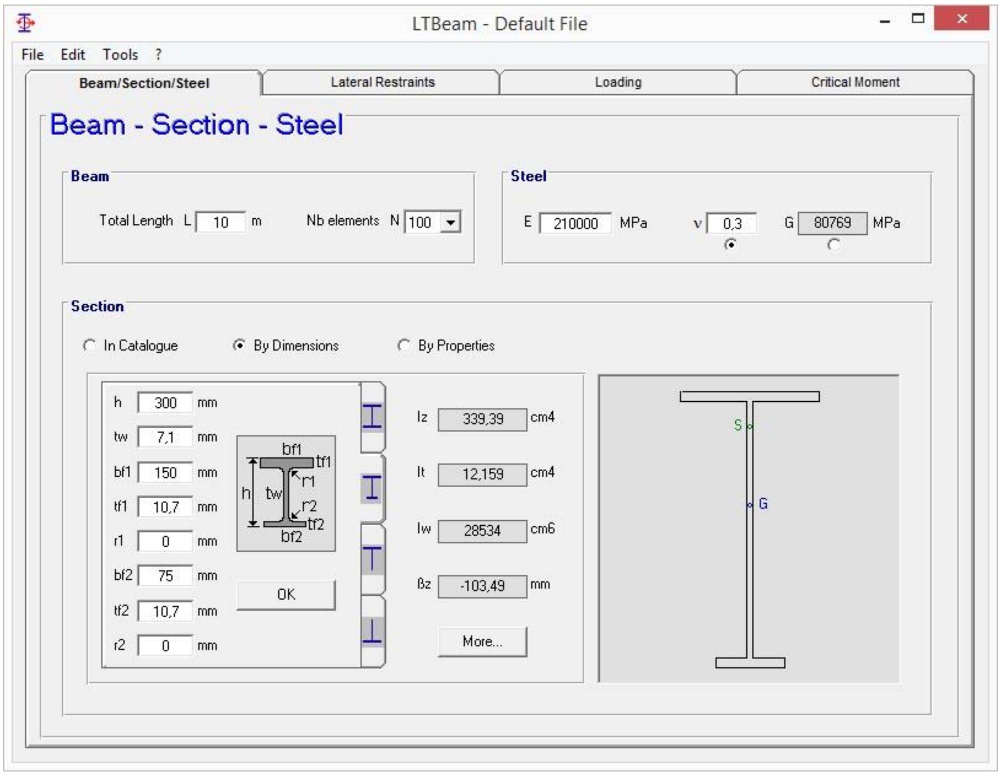



*Figure 3.7 User interface of the simple input mode in LTBeam.*

The software gives the possibility to choose members from its built-in catalogue, to manually enter the geometry or to enter the following sectional properties: The second moment of area about the minor axis $I_z$, the torsional constant $I_t$, the warping constant $I_w$ and the Wagner's coefficient $\beta_z$ (see Section 2.4.2.1). The last option must be used if beams with a channel section are to be modelled. No channel sections or monosymmetric I-sections are available from the built-in catalouge. Monosymmetric I-sections can be defined by choosing the **By Dimensions** option, as shown in Figure 3.7. Then the corresponding sectional properties will be calculated by the software.

The user can choose between 100, 120, 150 and 200 elements. When lateral restraints are chosen, fixed, free or spring restraints are the alternatives but for the end supports in the plane of bending, fixed and free are the only options. No spring supports are available in simple input mode. It is also a possibility to apply local lateral restraints or a continuous lateral restraint along the whole beam. Different types of loading can be applied; a point load, a distributed load and a point moment about the major axis. The location of the forces can be chosen along the beam and the point load and the distributed load can be placed at different heights as well. The cross section must be constant along the x-axis in the simple input mode. In order to model a multi-span beam, the interior supports in the plane of bending must be replaced by its reactions and applied as point loads.

### 3.3.2 File input mode

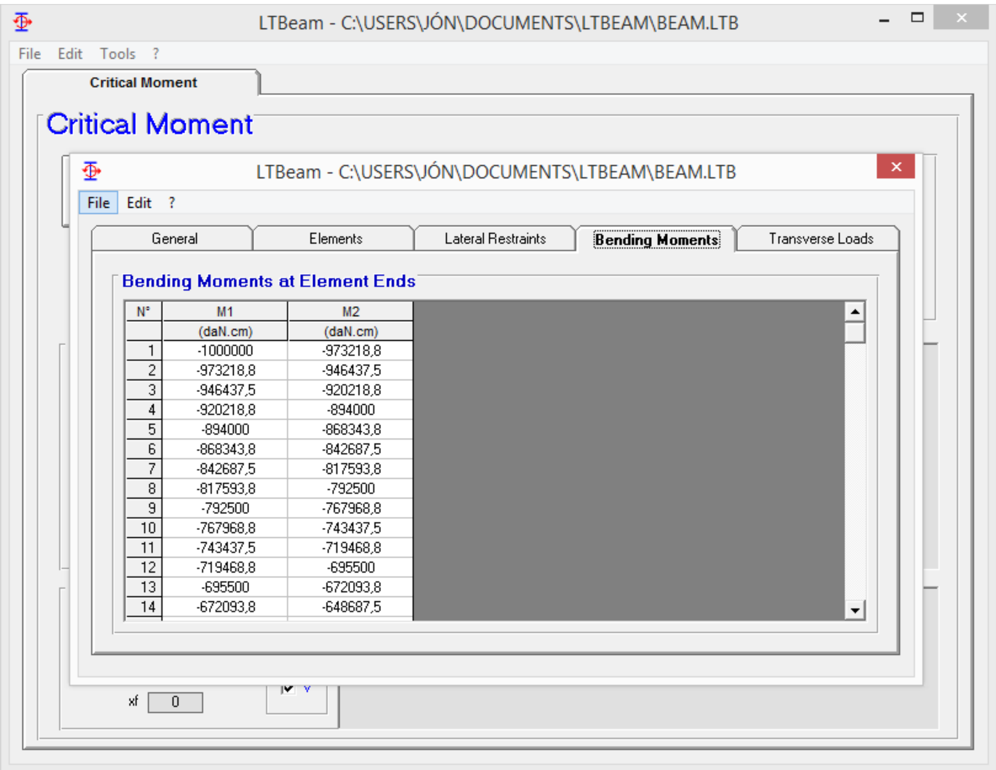



*Figure 3.8 User interface of the file input mode in LTBeam.*

To use the file input mode for more complex cases, a preparation and even pre calculations are sometimes needed. Here the cross section can vary, but the user must then define the sectional properties in each element. Between 50 and 300 elements can be chosen. The bending moment distribution for the whole beam has to be established and specified, it is defined by the values of bending moment in the endpoints of each element, as shown in Figure 3.8. Any loading, such as external moments and loads, and support reactions in the plane of bending are taken into account by the bending moment distribution. In addition to that, loads that are not applied at the shear centre of the beams cross section, and result in destabilising or stabilising effects, need to be specifically stated.

### 3.3.3 Elastic critical moment

When a beam reaches the buckling level, the behaviour is governed by the following relationship (CTICM 2012):

$$|K_L + \mu_{cr} \cdot K_G(\bar{M})| = 0 \tag{3.32}$$

Where $K_L$ = Linear matrix

$K_G$ = Geometrical matrix

$\bar{M}$ = Distribution of bending moment along the beam

$\mu_{cr}$ = Critical value of the multiplier $\mu$

The geometrical matrix $K_G$ is a function of certain dimensions and of the bending distribution $\bar{M}$ in the beam, resulting from the loading data. The linear matrix $K_L$ is a function of dimensions and material properties. The multiplier $\mu$ is used to multiply the load applied to the beam. $\mu_{cr}$, the critical value of $\mu$, is also called a eigenvalue. There are several eigenvalues, depending on the problem, but LTBeam uses the lowest positive one as $\mu_{cr}$ when continuing the process.

LTBeam determines the critical elastic moment $M_{cr}$ once $\mu_{cr}$ has been found by using the following formula (CTICM 2012):

$$M_{cr} = \mu_{cr} \cdot M_{max} \tag{3.33}$$

Where $M_{max}$ = Maximum moment value

$M_{cr}$ = Elastic critical moment

The determinant in equation (3.32) is positive in the stable domain but negative in the unstable domain. The eigenvector $X$, associated to $\mu_{cr}$, is found and the following applies (CTICM 2012):

$$[K_L + \mu_{cr} \cdot K_G(\bar{M})]|X| = 0 \tag{3.34}$$

Where $X$ = Eigenvector of the determinant

$X$ is a displacement vector which includes the lateral displacement, torsional rotation, lateral bending rotation and warping deformation.

## 3.4 SAP2000

The SAP name was introduced over 30 years ago (Computers & Engineering 2003). SAP2000 has therefore been evolving to a comprehensive software which can be used for structural analysis as well as in design. It has a 3D object based graphical environment, that can be seen in Figure 3.9, and can be used by designers working on problems regarding bridges, buildings, transportation and more.

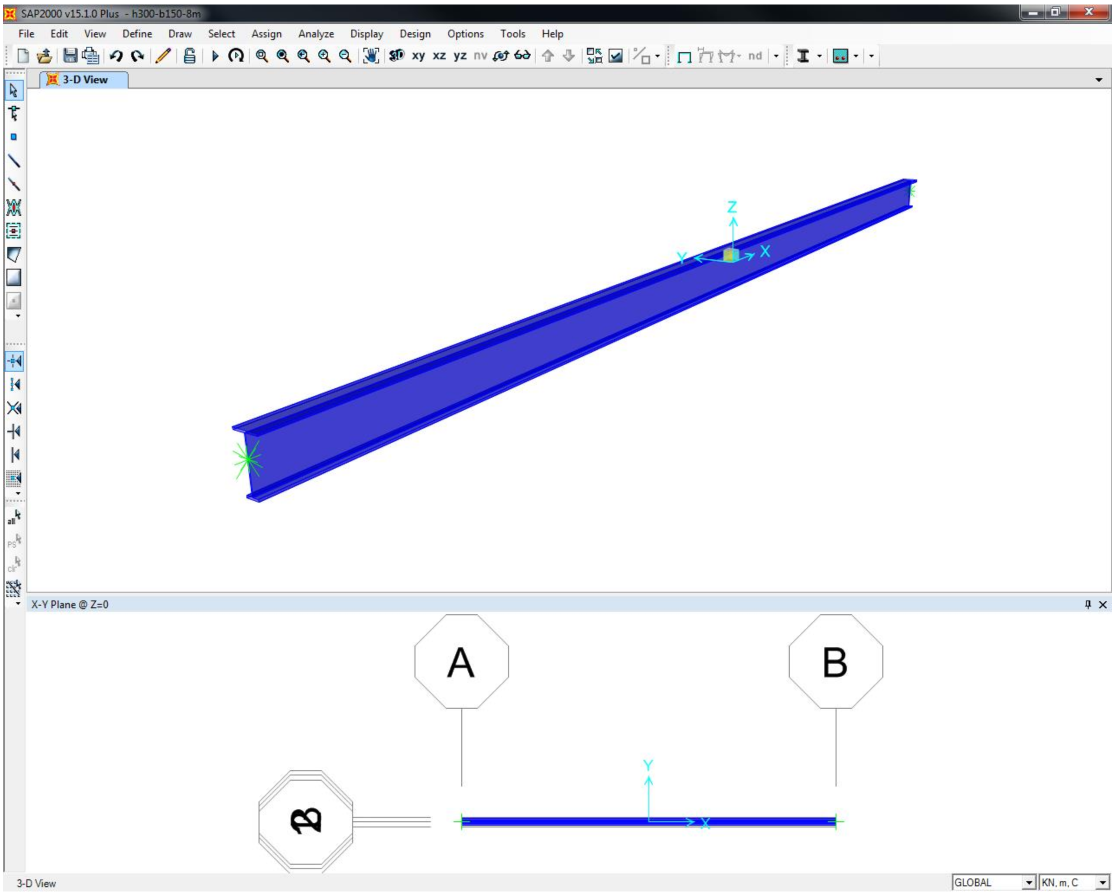



*Figure 3.9 The graphical interface of SAP2000 including a simple beam model.*

### 3.4.1 Elastic critical moment

SAP2000 has a design module. It can be used in design with regard to LT-buckling. The following equation is used to find $M_{cr}$ (Computers and Structures, Inc. 2011):

$$M_{cr} = C_1 \frac{\pi^2 E I_z}{L_{cr}^2} \left( \frac{I_w}{I_z} + \frac{L_{cr}^2 G I_t}{\pi^2 E I_z} \right)^{0.5} \tag{3.35}$$

Where $C_1$ = Factor depending on the moment diagram

$E$ = Young’s modulus of elasticity

$G$ = Shear modulus

$I_t$ = Torsional constant

| | | |
|---|---|---|
| $I_w$ | = | Warping constant |
| $I_z$ | = | Second moment of area about the minor axis |

The buckling length, $L_{cr}$, is defined as (Computers and Structures, Inc. 2011):

$$L_{cr} = k_z \cdot L \tag{3.36}$$

Where $L$ = Length of the beam between points which have lateral restraints

The effective length factor, $k_z$, that specifies the degree of end restraint about the minor axis, is set as 1 by default (Computers and Structures, Inc 2011, p 5-8). In the present thesis the degree of restraint for rotation about z-axis and warping is assumed to be the equal, which means $k_z = k_w$. The unsupported length for lateral-torsional buckling, $L$, is taken as the length between points that support the beam in the minor axis direction (Computers and Structures, Inc 2011, p 5-7).

The $C_1$ factor is found using the following method (Ahnlén & Westlund 2013) and (Arora & Wang n.d.):

$$C_1 = \frac{12.5M_{max}}{3M_2 + 4M_3 + 3M_4 + 2.5M_{max}} \tag{3.37}$$

See equation (3.26) and Figure 3.6 for description and illustration of variables.

Other descriptions of variables in equation (3.35) are as in equation (2.26). Comparing the equation used in SAP2000 to calculate $M_{cr}$, equation (3.35), with the full version of the 3-factor formula, equation (2.26), it can be seen that SAP2000 does not take explicitly into account the monosymmetry of sections or the vertical position of the load application.

The values for the parameters $k_z$, $L$ and $C_1$ in SAP2000 can also be manually overwritten.

## 3.5 STAAD.Pro

STAAD.Pro is a structural analysis and design engineering software released by Bentley Systems. The software uses FEM to perform a variety of analyses, such as linear static, response spectra, dynamic, buckling and non-linear geometric analysis. The software is also able to perform design code checks of structures for different types of material using number of design codes from around the world (Bentley Systems, Inc. 2014a & Bentley Systems, Inc. 2014b).

The structural model can be generated using graphical utilities in the software itself or by typing commands in a text editor. The software includes databases for standard structural elements, i.e. for standard cross sections and materials. Results can be viewed and verified by examining for example diagrams for displacements, moments and forces and contours for stresses and strains. Design checks can be made for structural members, connections and reinforcement (Bentley Systems, Inc. 2011).

All the features that STAAD.Pro offers, including those mentioned above, are integrated into one graphical interface, see Figure 3.10. On the ribbon above the structural model, the different modes can be chosen, such as the modelling mode and the post-processing mode. The rest of the graphical interface changes when changing between modes. In the upper part of the window the menus and toolbars are located, as is customary for engineering software. On the left side so called pages are located, that are a set of tabs, and each page has certain subpages. Each subpage has its own purpose in the mode specified and they are arranged in the reasonable order of operations. For example in the modelling mode the subpages are defined in the following order; job setup, definition of structural members, sectional properties, specifications, supports, loading, material and so on, in that order. The panel on the right is a data area that corresponds to the page chosen, where input data is entered (Bentley Systems, Inc. 2011).

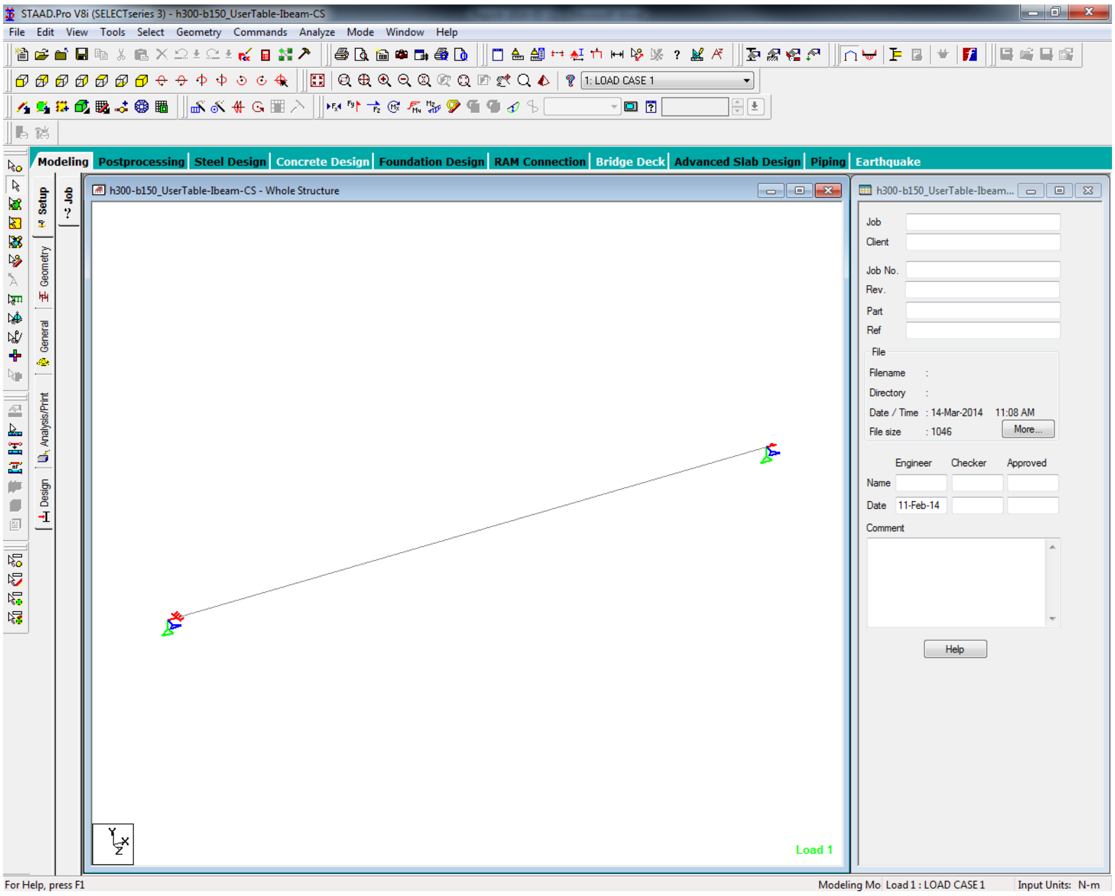



*Figure 3.10 The graphical interface of STAAD.Pro for the modelling mode. On the ribbon above the structural model are different modes, above that toolbars and menus. On the left are so called pages and data area on the right.*

This software is comprehensive and it may look overwhelming and confusing for a less advanced user. On the other hand arranging the pages in an order of operations helps making the modelling process more systematic.

### 3.5.1 Elastic critical moment

The elastic critical moment $M_{cr}$ is a parameter needed in order to perform Eurocode 3 design checks with regard to lateral-torsional buckling of beams subjected to bending, as is described in Section 2.5. For a steel beam model, STAAD.Pro can perform such design checks and obtains the parameters needed analytically. When using Eurocode 3 without a National Annex specified, STAAD.Pro uses the method specified in ENV 1993-1-1:1992 to evaluate $M_{cr}$, i.e. it uses the 3-factor formula given in equation (2.26). If a National Annex is specified STAAD.Pro uses the method given there, but if no method is specified the corresponding NCCI document for doubly symmetric and monosymmetric I-sections is used, that is Access Steel (2010) and Access Steel (2006). In these documents the 3-factor formula is also used, but with updated values for the C-factors. For other section types, where a National Annex with no specified method has been chosen, the method in ENV 1993-1-1:1992 will be used as long as it applies. For sections where none of the methods above apply, such as channel sections, STAAD.Pro will not calculate $M_{cr}$ in the design check. Instead it will use a method presented in Section 4.3.6 of the British Standard BS 5950-1:2000 to evaluate the buckling resistance moment $M_{b,Rd}$, without having to calculate $M_{cr}$ (Bentley Systems, Inc. 2011).

In order to perform the design check for lateral-torsional buckling, a beam model with the desired load case has to be established and a linear elastic analysis specified, by choosing **Perform Analysis** with a print all option under the analysis page. Under the design page in the modelling mode, design code checks can be made for steel members. Eurocode 3 is chosen as the design code by selecting EN 1993-1-1:2005 at the top of a command tree that appears in the data area on the right hand side, which is a summary of input file commands. An example of a command tree for a beam model is in Figure 3.11, where commands and design parameters can be seen. Then a special command is needed to actually perform the check, which can be added to the command tree by choosing **Commands**, under the command tree, and selecting **Check Code**. The command automatically appears with a question mark in front of it, which means the command still has to be assigned to the structural element by selecting the element and the command and choosing **Assign** at the bottom of the command tree data area.

A parameter folder should have appeared in the command tree, see Figure 3.11, where the appropriate parameters have to be added and assigned to the structure in order to perform the desired design check. By choosing **Define Parameters**, values for the available parameters can be specified and the parameters added to the input file. In order for $M_{cr}$ to be displayed in the STAAD.Pro output file it is crucial to define the parameter **TRACK** to give detailed results.

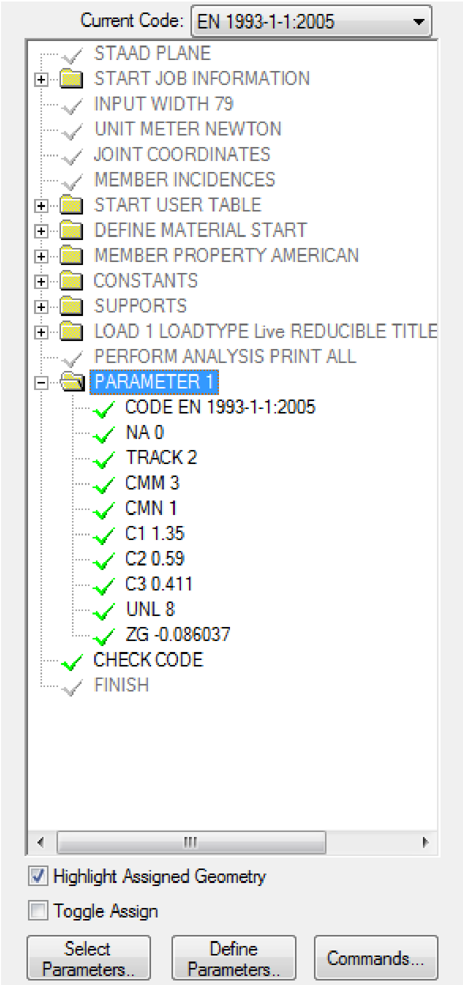



*Figure 3.11 Example of a command tree for steel design in STAAD.Pro, with the relevant design parameters.*

STAAD.Pro is not able to collect the necessary inputs to the 3-factor formula from the beam model itself, but they have to be defined through the design parameters, specified in Table 3.1. The load case modelled will not be recognised, so the C-factors can be chosen by defining the parameter **CMM**, and then they will be selected from a table in the pre-standard ENV 1993-1-1:1992 if Eurocode 3 without National Annex is used. The C-factors can also be manually defined with the parameters **C1**, **C2** and **C3**, and STAAD.Pro will use the values from these parameters if they are specified, even though **CMM** is also defined. The degree of lateral end-restraint can be defined with the parameter **CMN** in the form of an effective length factor, where it is assumed that $k_z$ is equal to $k_w$, see Section 2.1.8. It is also possible, and in some cases it can lead to more reliable results in the authors' experience, to insert the unrestrained length $k \cdot L$, where $k = k_z = k_w$, manually as the parameter **UNL**. The point of load application in the vertical direction is defined with the parameter **ZG**, which is equivalent to the variable $z_g$ in the 3-factor formula, see equation (2.26). All defined commands have to be assigned to the structure, as described above, and assigned commands acquire a checkmark as can be seen in Figure 3.11.

*Table 3.1 Relevant parameters in STAAD.Pro concerning design check for lateral-torsional buckling (adopted from Bentley Systems, Inc. 2011).*

| Parameter name | Description |
|---|---|
| CODE | Should be specified as EN 1993-1-1:2005 to perform a design check according to Eurocode 3. |
| C1 | Manual input of the $C_1$ factor in the 3-factor formula, see equation (2.26). |
| C2 | Manual input of the $C_2$ factor in the 3-factor formula, see equation (2.26). |
| C3 | Manual input of the $C_3$ factor in the 3-factor formula, see equation (2.26). Works only for T-sections. |
| CMM | Type of loading and support conditions. If the parameters C1, C2 and C3 are not specified, this parameter controls the choice for $C_1$, $C_2$ and $C_3$ through a table with load cases in the supplementary document for the chosen design method. |
| CMN | Degree of end-restraint against rotation about the vertical axis and warping, in the form of effective length factors, see Section 2.1.8. |
| NA | Choice of National Annex. If the plain Eurocode 3 is to be used nothing is selected. |
| TRACK | Level of detail for results displayed in the output file. $M_{cr}$ will be displayed if the parameter is set to **Detailed results**. |
| UNL | Manual input of the unrestrained length of the beam $k \cdot L$ if $k = k_z = k_w$, see Section 2.1.8. |
| ZG | Coordinate of the point of load application w.r.t. shear centre in z-direction; see the variable $z_g$ in equation (2.26). |

The analysis is performed by expanding the **Analyze** tab and choosing **Run Analysis**. When the analysis is successfully finished, an option to **View Output File** can be chosen, but the output can also be opened from the toolbar area. $M_{cr}$ can be found under **Buckling Calculations** in the output file, as can be seen in Figure 3.12.

```
BUCKLING CALCULATIONS (units - kN,m)
   Lateral Torsional Buckling Moment       MB =   57.8
   co-efficients C1 & K : C1 =1.350 K =1.0, Effective Length= 8.000

   Elastic Critical Moment for LTB,              Mcr   =    57.8
   Critical Load For Torsional Buckling,         NcrT  =   852.3
   Critical Load For Torsional-Flexural Buckling, NcrTF =   852.3
```

*Figure 3.12 Example of a part of an output file in STAAD.Pro when performing steel design checks. The resulting elastic critical moment $M_{cr}$ for lateral-torsional buckling can be seen.*

STAAD.Pro does not yet recognise monosymmetric I-sections when evaluating $M_{cr}$ and the **C3** parameter is only taken into account when designing T-sections. Therefore no factor accounts for monosymmetry of a section and the software should give less accurate results for $M_{cr}$ of beams with monosymmetric I-sections. Since the 3-factor formula does not apply for beams with channel sections, STAAD.Pro does not calculate $M_{cr}$ for such beams (Bentley Systems, Inc. 2011).

# 4 Software Application Study

## 4.1 Cases studied

Two types of cross sections will be studied for a single spanned beam, supported at both ends. Setup configuration varies for both cases. The dimensions of the studied cross sections are chosen to match beams from published papers, where reference values for $M_{cr}$ could be found; Mohri et al. (2003) for the beam with a monosymmetric I-section and Snijder et al. (2008) for the beam with a channel section. The cross sections and other configurations of the beams are introduced below.

### 4.1.1 Monosymmetric I-beam

The beam with the monosymmetric I-section chosen to be modelled will be made of elastic material, with $E = 210$ GPa and $\nu = 0.3$. The beam length is $L = 8$ m. Hinged end supports will be used, i.e. $k_y = 1$ (see Section 2.1.8), with varied degree of end restraint for the rotation about the z-axis and warping. The beam will be subjected to a concentrated load at mid-span, with varied load height.

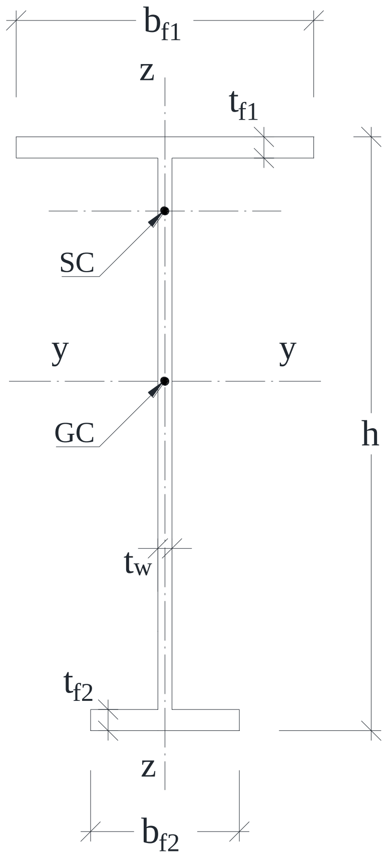



*Figure 4.1 The monosymmetric I-section that will be used in the study, in the right proportions.*

The dimensions of the chosen monosymmetric I-beam cross section are illustrated in Figure 4.1 and are as follows:

- $h = 300$ mm
- $b_{f1} = 150$ mm
- $b_{f2} = 75$ mm
- $t_{f1} = t_{f2} = 10.7$ mm
- $t_w = 7.1$ mm

The sectional parameters needed for calculations and modelling are as follows, where equations (2.9), (2.16) and (2.33) were used to calculate $I_t$, $I_w$ and $z_j$ respectively:

- $A = 4.386 \cdot 10^3$ mm$^2$
- $I_y = 6.012 \cdot 10^7$ mm$^4$
- $I_z = 3.394 \cdot 10^6$ mm$^4$
- $I_t = 1.251 \cdot 10^5$ mm$^4$
- $I_w = 2.805 \cdot 10^{10}$ mm$^6$
- $z_j = 103.77$ mm

Two separate studies will be performed using the monosymmetric I-beam described above. In the former study the concentrated load will be applied at the centre of gravity of the beams mid-section, so $z_g = -86.04$ mm. The lateral position of the load will not be varied, i.e. $y_g = 0$, so the load will be applied centrically. The degree of the end restraint for rotation about the z-axis and warping will be kept equal, but varied as follows (see illustration in Figure 4.2):

- $k_z = k_w = 1$
- $k_z = k_w = 0.7$
- $k_z = k_w = 0.5$

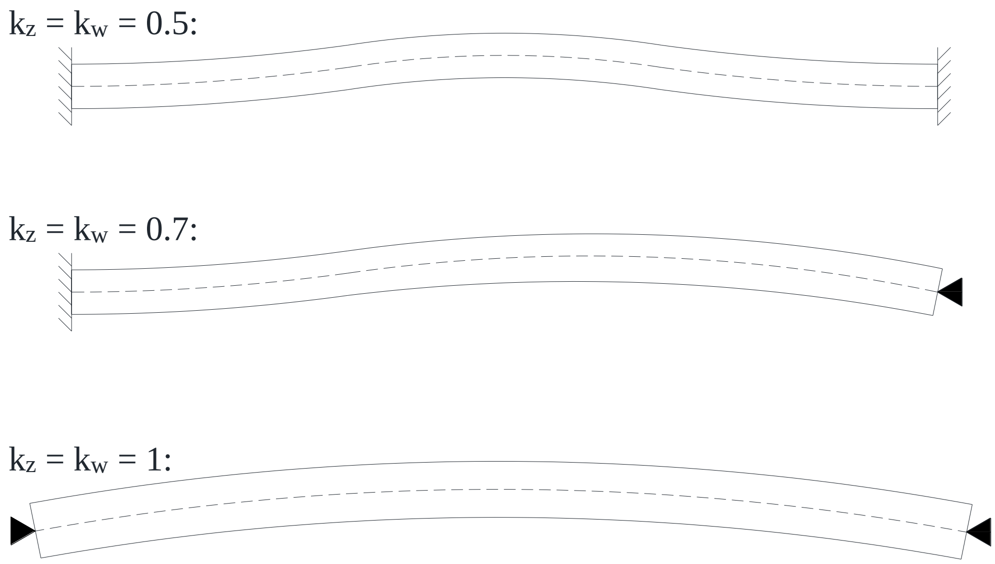



*Figure 4.2 Illustrative comparison of varied degree of end restraint for rotation about the z-axis and warping, for a beam seen from above.*

A comparison will be made of the resulting $M_{cr}$ between the following six calculation methods:

- ADINA; performing a linearized buckling analysis using warping beam elements with a general cross section and sectional properties as input, modelled as described in Section 4.2.5.
- Colbeam; using a welded cross section with dimensions as input, modelled as described in Section 4.2.1.
- LTBeam; using dimensions as input, modelled as described in Section 4.2.2.
- SAP2000; using an I-section with dimensions as input, modelled as described in Section 4.2.3.
- STAAD.Pro; using a general cross section with sectional properties as input, modelled as described in Section 4.2.3.
- 3-factor formula; using equation (2.26) with the C-factors from Table 2.1.

The exact values for the sectional parameters listed above are not used in all calculation methods, and numerical comparison of the values used in each method can be seen in Table 4.4.

In the latter study the degree of end restraint will be specified as $k_z = k_w = 1$. The lateral position of the concentrated mid-span load will not be varied, i.e. $y_g = 0$, so the load will always be applied centrically, but the vertical position of the PLA for the load will be varied as follows (see illustration in Figure 4.3):

- Top flange, i.e. the top surface of the section, with $z_g = -37.49$ mm
- Shear centre, with $z_g = 0$ mm
- Centre of gravity, with $z_g = -86.04$ mm
- Bottom flange, i.e. the bottom surface of the section, with $z_g = 262.51$ mm

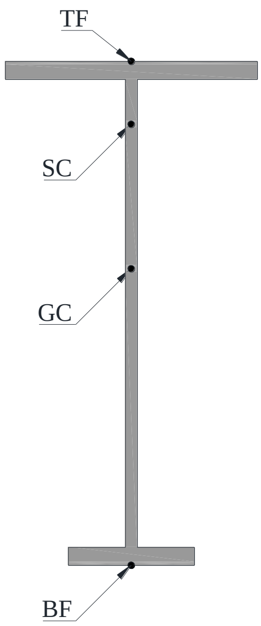



*Figure 4.3 Illustrative comparison of the different vertical positions of the PLA for the monosymmetric I-beam. TF stands for the top flange, SC for the shear centre, GC for the centre of gravity and BF for the bottom flange.*

A comparison will be made of the resulting $M_{cr}$ between the following six calculation methods:

- ADINA; performing a linearized buckling analysis using warping beam elements with a general cross section and sectional properties as input, modelled as described in Section 4.2.5.
- Colbeam; using a welded cross section with dimensions as input, modelled as described in Section 4.2.1.
- LTBeam; using dimensions as input, modelled as described in Section 4.2.2.
- SAP2000; using an I-section with dimensions as input, modelled as described in Section 4.2.3.
- STAAD.Pro; using a general cross section with sectional properties as input, modelled as described in Section 4.2.3.
- 3-factor formula; using equation (2.26) with the C-factors from Table 2.1.

The exact values for the sectional parameters listed above are not used in all calculation methods, and numerical comparison of the values used in each method can be seen in Table 4.4.

In addition to that the results from the study of varied height of the PLA will also be compared to reference values given by Mohri et al. (2003).

### 4.1.2 Channel beams

The beams with the channel section chosen to be modelled will be made of an elastic material, with $E = 210$ GPa and $\nu = 0.3$. The beam length will be varied. Hinged end supports will be used, more specifically the fork supports (see Section 2.1.8), with constant degree of end restraint for the rotation about z-axis and warping, i.e. $k_y = k_z = k_w = 1$. Two types of load will be separately applied to the beams. The lateral position of the load will be varied, so the load will both be applied centrically and eccentrically.

The dimensions of the chosen channel cross section are illustrated in Figure 4.4 and are as follows:

- $h = 160$ mm
- $b_f = 70$ mm
- $t_f = 10$ mm
- $t_w = 6.5$ mm

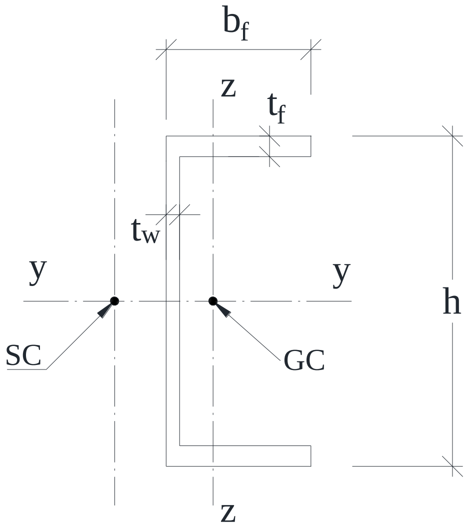



*Figure 4.4 The channel cross section that will be used in the study, in the right proportions.*

The sectional properties needed for calculations and modelling are as follows, where equations (2.12), (2.20), (2.33) and (2.3) were used to calculate $I_t$, $I_w$, $z_j$ and $e$ respectively:

- $A = 2.310 \cdot 10^3$ mm$^2$
- $I_y = 9.373 \cdot 10^6$ mm$^4$
- $I_z = 1.131 \cdot 10^6$ mm$^4$
- $I_t = 5.823 \cdot 10^4$ mm$^4$
- $I_w = 4.426 \cdot 10^9$ mm$^6$
- $z_j = 0$ mm
- $e = 26.81$ mm

The cross section is a modified version of the rolled channel section UPE160, adjusted for modelling with shell elements, by neglecting the radius of the rolled section and instead increasing the thicknesses of the cross section parts.

Two separate studies will be performed using the channel beam described above. In the former study two conditions will be varied in two ways each; the length of the beam and the type of loading, and all four possible cases will be considered. Both types of loading will be acting at the shear centre, i.e. $y_g = z_g = 0$, so the load will be applied centrically.

The length of the beam will be varied as follows:

- $L = 2.8$ m
- $L = 4$ m

The types of loading used to apply on the beam are as follows:

- A concentrated load at mid-span.
- A uniformly distributed load over the whole span.

A comparison will be made of the resulting $M_{cr}$ between the following five calculation methods:

- ADINA; performing a linearized buckling analysis using shell elements, modelled as described in Section 4.2.5.
- Colbeam; using a rolled UPE160 cross section, modelled as described in Section 4.2.1.
- LTBeam; using sectional properties as input, modelled as described in Section 4.2.2.
- SAP2000; using a channel section with dimensions as input, modelled as described in Section 4.2.3.
- 3-factor formula; using equation (2.26) with the C-factors from Table 2.1.

The exact values for the sectional parameters listed above are not used in all calculation methods, and numerical comparison of the values used in each method can be seen in Table 4.5.

The software STAAD.Pro will not be used, since it does not offer a method to evaluate $M_{cr}$ for beams with channel sections (see Section 3.5.1).

In the latter study the same four cases will be considered as in the former one, i.e. using two different lengths for the beam, $L = 2.8$ m and $L = 4$ m, and two types of loading, a concentrated mid-span load and a uniformly distributed load. Both types of loading will be acting at the height of the shear centre, i.e. $z_g = 0$. The lateral position of the load will be varied as follows (see illustration in Figure 4.5):

- $y_g = 0$ mm (shear centre)
- $y_g = 5$ mm
- $y_g = 10$ mm
- $y_g = 26.81$ mm (centre of web)

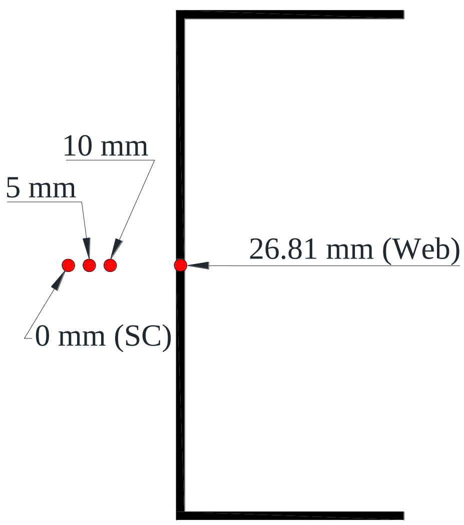



*Figure 4.5 Illustrative comparison of the different load eccentricities, $y_g$, used for the channel beams. The dots represent the lateral position of the load for each eccentricity with respect to the section of the channel beam.*

Comparison will be made of the resulting maximum bending moment in the beam at different level of loading for the different lateral position of the load, in order to evaluate if a critical value of the moment can be observed for eccentric loading. The commercial software ADINA will be used to perform a collapse analysis with the LCD method using shell elements and initial geometric imperfections, modelled as described in Section 4.2.6.

In addition the resulting $M_{cr}$ obtained for the corresponding centrically loaded case in the former study by performing a linearized buckling analysis in ADINA will be compared to the results from the incremental collapse analysis.

## 4.2 Modelling methods

The following sections present details of the modelling methods, that were conducted using the different commercial structural engineering software introduced in Chapter 3.

### 4.2.1 Modelling in Colbeam

The member design software Colbeam is introduced in Section 3.2 and its user interface can be seen in Figure 3.3. The choice between the materials Colbeam offers does not matter in the evaluation of $M_{cr}$ and therefore the first thing to do would be to define the cross section. The monosymmetric I-section, defined in Section 4.1.1, can be chosen under the category of **Welded** cross sections, where the dimensions must be manually inserted, as can be seen in Figure 4.6. Under **Rolled** cross sections the standard UPE version of the channel section defined in Section 4.1.2 is chosen as UPE160, as showed in Figure 4.7. In both cases the corresponding sectional properties are calculated by the software.

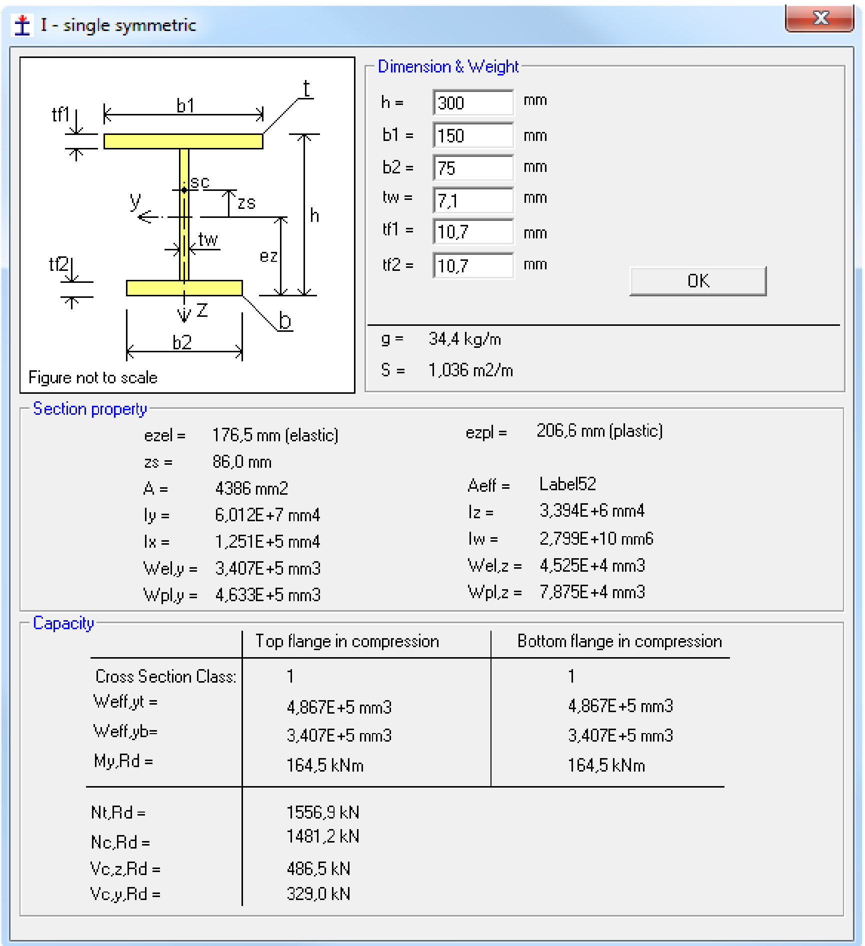



*Figure 4.6 Dimensions input window for monosymmetric I-section in Colbeam, with the corresponding sectional properties. It should be noted that the sectional property **Ix** is the torsional constant $I_t$.*

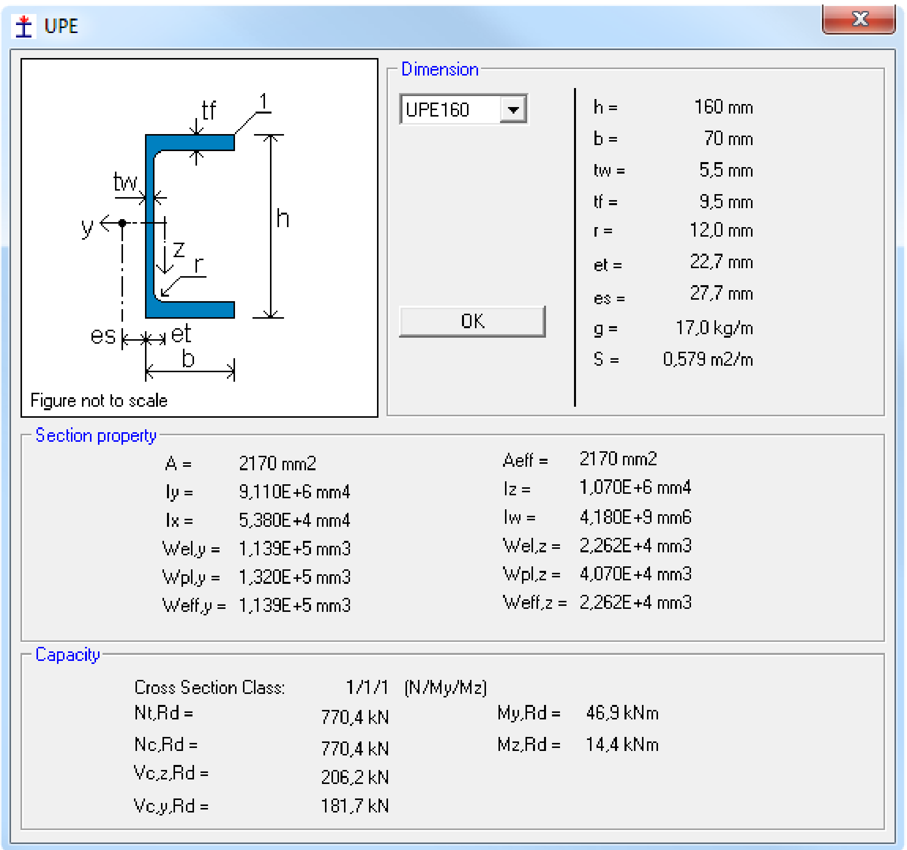



*Figure 4.7 Dimensions input window for UPE channel section in Colbeam, with the corresponding sectional properties. It should be noted that the sectional property* ***Ix*** *is the torsional constant $I_t$.*

The boundary condition at the vertical support, i.e. the degree of restraint for the translation in z-direction and the rotation about y-axis in each end, are chosen under the **Support** area in the main window, see Figure 3.3. In the **Member data** area the length of the member is specified. Under the **Loading** tab the desired load is defined, with distributed load noted as **q-load** and point load as **P-load**. The magnitude applied does not matter when evaluating $M_{cr}$, but some value must be inserted for the software to acquire the shape of the moment diagram. For the loads the direction must be but as **Z** and the position specified as half the span for the point load.

In the **Lateral Torsional Buckling** area the corresponding option is selected, and under **Input** the relevant parameters are specified, see Figure 4.8. The **Lateral torsional buckling length** is specified as $k = k_w = k_z$, see Section 2.1.8, i.e. this parameter includes both the degree of restraint for rotation about z-axis and warping for the beam. Under **Load level** the position of PLA in the z-direction is specified, see Section 2.1.7, either with the included shortcuts for the top, middle and bottom of the cross section or by directly inserting the $z_a$ variable. The C-factors are specified under **Lateral torsional bending/buckling capacity**, either by using the built-in, tabulated values or by **Manual input of C-factors**, as described in Section 3.2.1.

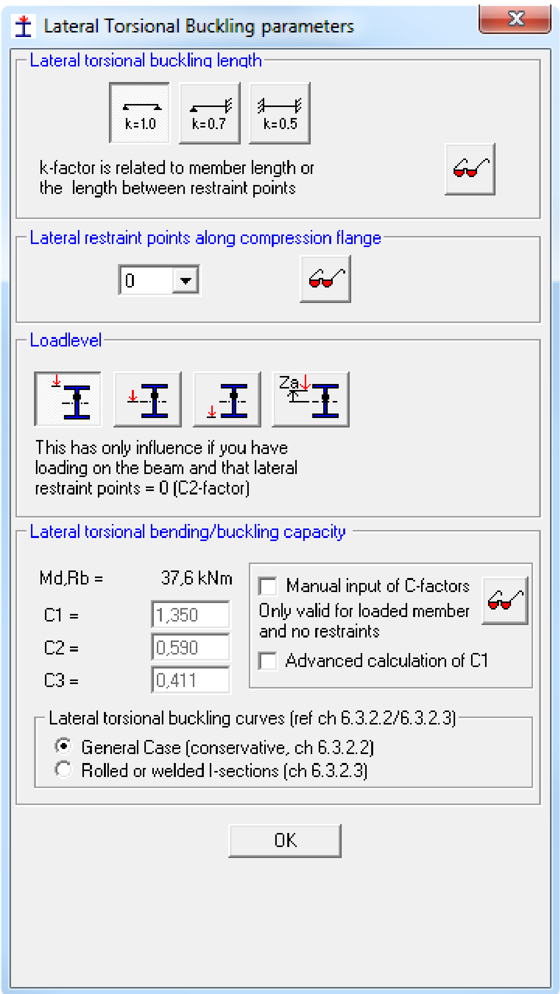



*Figure 4.8 Lateral-torsional buckling parameters window in Colbeam.*

## 4.2.2 Modelling in LTBeam

The FEM based $M_{cr}$ calculation tool LTBeam is described in Section 3.3. The modelling is performed by using the simple input mode (see Section 3.3.1).

In the **Beam/Section/Steel** tab the length of the beam is set to relevant value, depending on the case (see Sections 4.1.1 and 4.1.2). Number of elements is set to 200, which gives the most accurate results. The Young's modulus of elasticity and the Poisson's ratio are chosen as the desired values. In the **Section** box the cross section is chosen. For the monosymmetric I-section the **By dimensions** option is used to define the dimensions of the cross section. The dimensions can be found in Figure 4.9. LTBeam doesn't give any option of putting in dimensions of channel sections. Instead the **By Properties** option must be used and the values for the sectional parameters $I_z$, $I_t$, $I_w$ and $\beta_z$ inserted. The input values are calculated by hand and can be found in Section 4.1.2. It should be noted, that the monosymmetry parameter $\beta_z = 0$ for the channel beams as they are symmetric about the major axis.

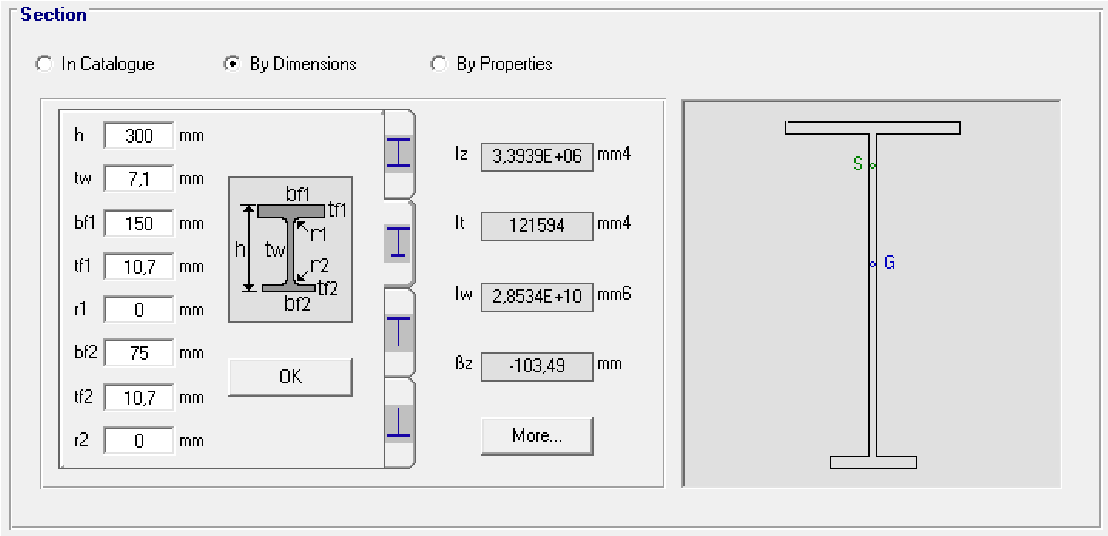



*Figure 4.9* *Input values in LTBeam for the monosymmetric I-beam used in the present thesis.*

In the **Lateral Restraints** tab only the **Left End** and **Right End** boxes are relevant in the present thesis. The choices between fixed/free are presented in Table 4.1, and how to obtain different values of the effective buckling length factors $k = k_w = k_z$ for a beam.

*Table 4.1* *Settings of the parameters v, θ, v' and θ' to obtain different values of the effective buckling length factors $k = k_w = k_z$ for a beam.*

| Parameter | Description | $k_w = k_z = 1$ | $k_w = k_z = 0.7$ | $k_w = k_z = 0.5$ |
|---|---|---|---|---|
| v | z-translation (affects $k_z$) | Fixed at both ends | Fixed at both ends | Fixed at both ends |
| $\theta$ | x-rotation | Fixed at both ends | Fixed at both ends | Fixed at both ends |
| v' | z-rotation (affects $k_z$) | Free at both ends | Fixed at one end, free at one end | Fixed at both ends |
| $\theta'$ | Warping (affects $k_w$) | Free at both ends | Fixed at one end, free at one end | Fixed at both ends |

In the **Loading** tab, the degree of end restraint for rotation about y-axis is specified as in Figure 4.10.

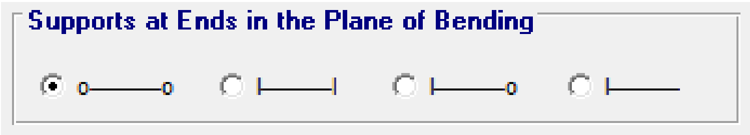



*Figure 4.10 The illustration used in LTBeam to represent different degree of end restraint for rotation about y-axis and translation in z-direction. From left, the choices are $k_y = 1$, $k_y = 0.5$, $k_y = 0.7$ and $k_y = 2$ (cantilever beam).*

When the loading is determined, the parameters in the **Loading** tab are chosen as in Table 4.2.

*Table 4.2 Settings of the parameters when a beam undergoes uniformly distributed load or point load.*

| Parameter | Description | Set to |
|---|---|---|
| q1 | Value of the distributed load at the left end of the beam seen from user. | Negative value, the load acts in negative z-direction. |
| q2 | Value of distributed load at the right end of the beam seen from user. | Same as q1 to obtain uniformly distributed load. |
| F | Value of point load. | Negative value, the load acts in negative z-direction. |
| xf1 | Location of q1 (relative distance from left end to right end). | 0 (left end of beam) |
| xf2 | Location of q2 (relative distance from left end to right end). | 1 (right end of beam) |
| xf | Location of F (relative distance from left end to right end). | 0.5 (at the centre of the span) |
| z /S | Vertical position of the loading. Coordinates in z-direction with an origin at the shear centre. | Relevant value, which varies between cases. |

To obtain results for $M_{cr}$ the **Proceed** button is pressed in the **Critical Moment** tab.

### 4.2.3 Modelling in SAP2000

When modelling the beams used in the current thesis using SAP2000, the **Beam** template is used and the units can be set as **N, m, C**. Number of spans are 1 and length is set to the relevant value.

In the **Define Materials** window the steel is chosen by pressing **Add New Material** and choosing European steel using standard **EN 1993-1-1 per EN 10025-2**. This gives $E = 210\ \text{GPa}$. Other values are not relevant for the LT-buckling calculations in SAP2000.

To be able to define monosymmetric I-sections and channel cross sections, the **Add New Property** option must be used in the **Frame Properties** window. The momosymmetric I-section can be created using the **I / Wide Flange** category and the sectional properties can be inserted as shown in Figure 4.11.

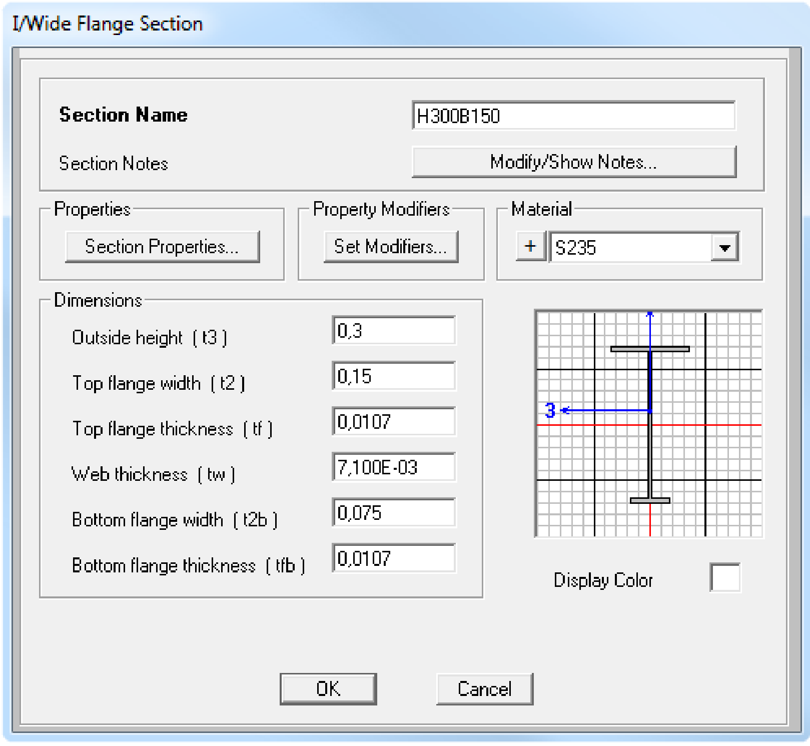



*Figure 4.11 Input values for sectional properties of the monosymmetric I-section used in SAP2000.*

When defining the channel cross section the **Channel** category can be used and the sectional properties can be inserted as shown in Figure 4.12.

In the **Frame Insertion Point** window the **11 (Shear Center)** option is chosen as the **Cardinal Point**. By doing so, the position of the cross section on the longitudinal axis of the beam is set as the shear centre, so all loads and supports that are applied on the beam will act in that point of the section. This does not affect the calculation of $M_{cr}$.

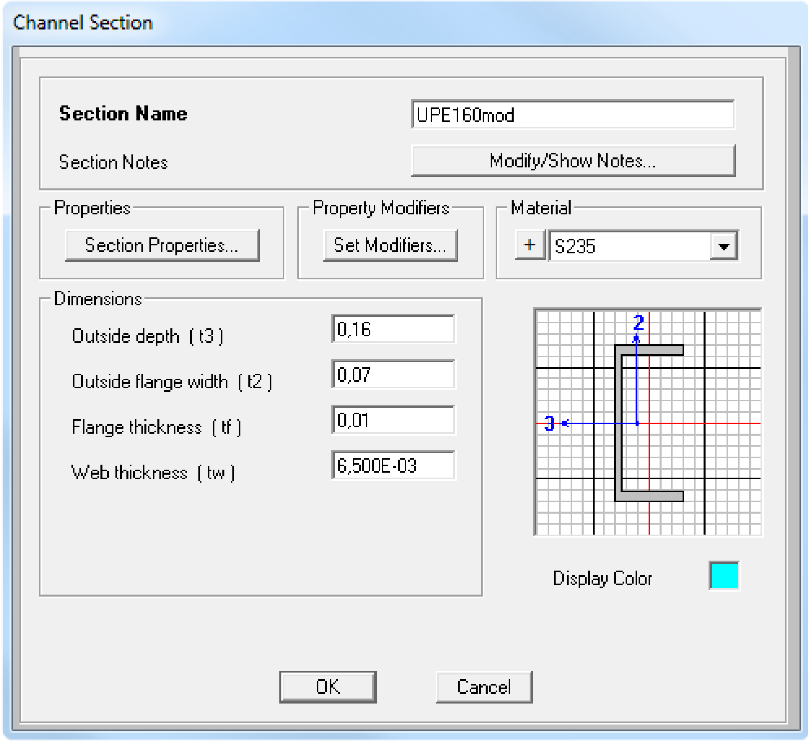



*Figure 4.12* *Input values for sectional properties of the channel section used in SAP2000.*

When defining the end supports, the desired fixed degrees of freedom are selected for each support in the **Joint Restraints** window. The hinged supports (see Section 2.1.8) are defined as shown in Figure 4.13 for one end, but the x-translation noted as **Translation 1** is left free for the other end.

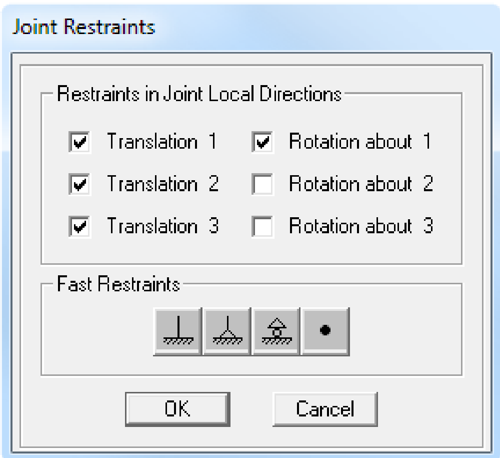



*Figure 4.13* *Definition of a hinged support in SAP2000 for one end of the beam, but at the other the **Translation 1** is kept free.*

Point load is applied using the **Frame Point Loads** option and distributed load using the **Frame Distributed Loads** option. The direction of the load should be **Gravity** and the value inserted as positive. The point load is put at mid-span, by specifying the relative distance from either end as 0.5, and the distributed load by specifying the magnitude of the load under **Uniform Load**. When defining the loads, the **Load Pattern Name** is chosen as a load pattern that has the **Self Weight Multiplier** set as zero, but the load pattern can be defined by clicking on the plus next to it.

A load case should be defined with the **Load Case Type** as **Static** and the **Analysis Type** as **Linear**, as shown in Figure 4.14. The load pattern used when defining the load is specified and its **Scale Factor** set to 1.

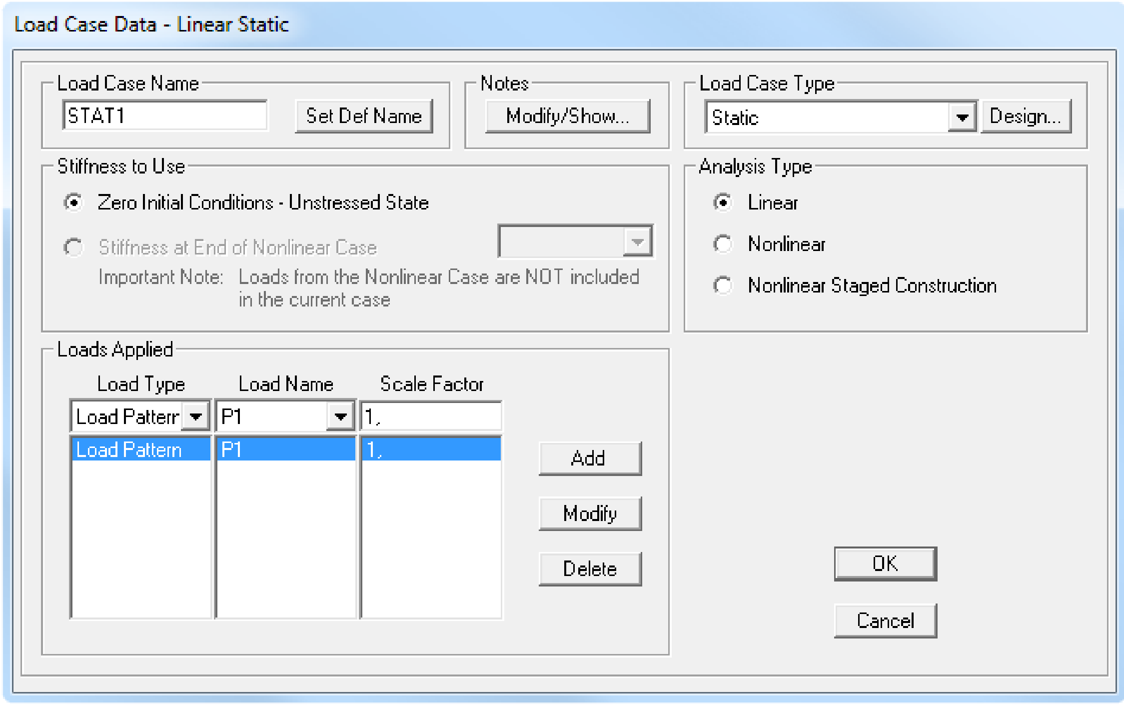



*Figure 4.14 Definition of a linear static load case in SAP2000.*

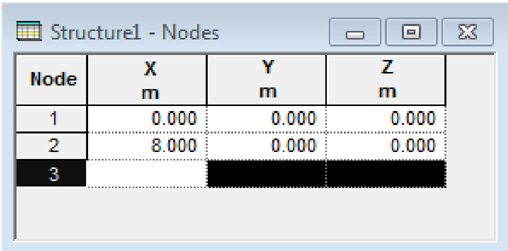
As mentioned in Section 3.4.1 the design parameters $k_z$, $L$ and $C_1$ that effect the calculation of $M_{cr}$ can be manually overwritten. This can be done before or after running the analysis in the **Steel Frame Design Overwrites for Eurocode 3-2005** window by specifying values for the **Effective Length Factor (K LTB)**, the **Unbraced Length Ratio (LTB)** and **Bending Coefficient (C1)** respectively. In the current study only the $k_z$ parameter is overwritten, with the corresponding value.

To perform the analysis, the **Run Analysis** button is clicked, the **Action** of the linear static load case previously defined is set to **Run** before pressing the **Run Now** button. The **Start Steel Design/Check of Structure** option is used to perform a design check of the structure. By right clicking on the beam a **Steel Stress Check Information** window opens, where **Summary** under **Display Details for Selected Item** can be specified. The **Steel Stress Check Data** window opens where $M_{cr}$ as well as other design values can be viewed.

## 4.2.4 Modelling in STAAD.Pro

When STAAD.Pro is launched with the intention of modelling a beam and evaluating its $M_{cr}$, **Eurocodes** must be checked under **License Configuration** before initiating a new project. A **Space** type of structure is chosen in the **New** window that appears and the length and force units specified as meters and newtons is recommended. On the next window choose to **Add Beam** to go to the modelling mode. The pages and subpages of the modelling mode, introduced in Section 3.5, will be covered when modelling the monosymmetric I-beam, defined in Section 4.1.1.

The modelling mode opens on the **Beam** subpage where the endpoints of the beam are specified by clicking on coordinates (0,0) and (8,0) on the XY-grid that appeared, for a beam length of 8 meters. The data area on the right shows the nodes existing, as can be seen in Figure 4.15, where the nodes also could have been manually created.




| Node | X m | Y m | Z m |
|---|---|---|---|
| 1 | 0.000 | 0.000 | 0.000 |
| 2 | 8.000 | 0.000 | 0.000 |
| 3 | | | |

*Figure 4.15 Data area of the **Beam** subpage in the modelling mode in STAAD.Pro where an example of created nodes in a beam model can be seen.*

The monosymmetric I-section must be created manually expanding **Tools** in the menu bar and selecting **Create User Table**. After clicking **New Table** a section type must be selected. It is possible to create a monosymmetric I-section with the type **ISECTION**, but in order to be able to perform design checks the section type must be taken as **GENERAL** (Bentley Systems, Inc. 2011). After selecting the section type, its properties must be specified by choosing **Add New Property**. The profile of the section can be defined by specifying the z- and y-coordinates of all its corners with the option **Define Profile Polygon**. Then its sectional properties would be computed by the software, but it is not provided what formulas are used for the computations. A more convenient way here is a manual input of the sectional properties, which provides a better comparison between programs. Description of the sectional properties that need to be assigned can be seen in Table 4.3 and the values used for the monosymmetric I-section, described in Section 4.1.1, can be seen in Figure 4.16.

*Table 4.3 Relevant sectional properties in STAAD.Pro needed to create a **General** cross section via **User Table** for a beam, which $M_{cr}$ should be evaluated for (adopted from Bentley Systems, Inc. 2011).*

| Property | Description |
|---|---|
| Ax | Area of cross section, $A$. |
| D | Depth of section, $h$. |
| TD | Thickness of sectional part parallel to depth (usually web), $t_w$. |
| B | Width of section, $b$. |
| TB | Thickness of sectional part parallel to flange, $t_f$. |
| Iz | Second moment of area about the major axis, $I_y$. |
| Iy | Second moment of area about the minor axis, $I_z$. |
| Ix | Torsional constant, $I_t$. |
| HSS | Warping constant, $I_w$. |
| DEE | Depth of web (or distance between fillets), $h_w$. |

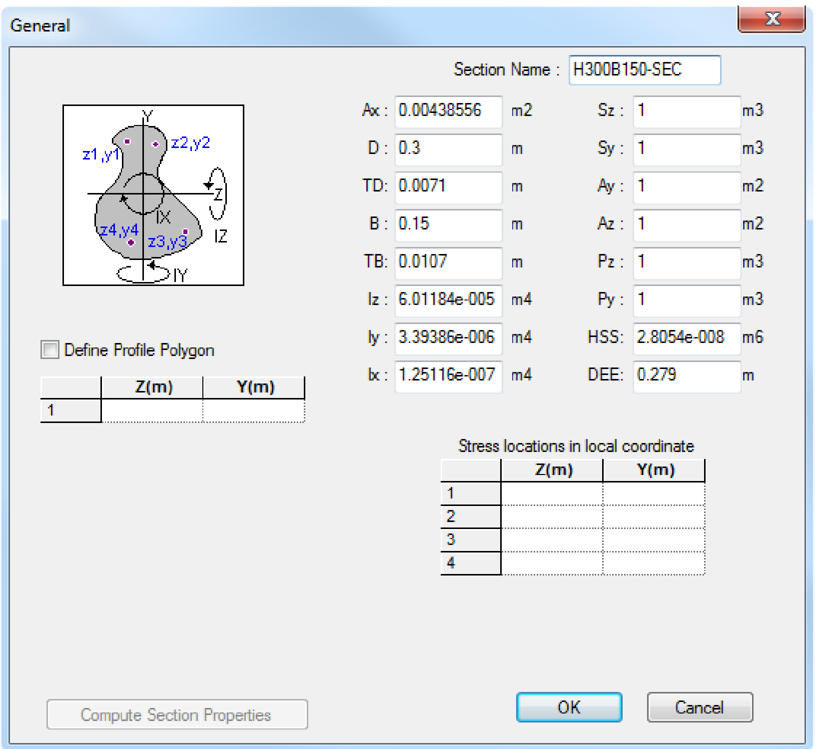



*Figure 4.16 Values of sectional properties used when defining the monosymmetric I-section in STAAD.Pro.*

A built-in steel material exists in STAAD.Pro, but not with the desired modulus of elasticity. Therefore a new material is created under the **General** page and **Material** subpage. By selecting **Create** on the data area on the right hand side, the desired value for the modulus of elasticity can be inserted and no more material parameters need to be defined.

Since the cross section has been created, the **Property** subpage can be chosen and then **User Table** on the data area. The cross section that was created is selected along with the material and added and assigned to the beam, in the same way as described in Section 3.5.1.

On the **Support** subpage the supports can be defined, by selecting **Create** on the data area and then **Fixed But**. The translational degrees of freedom are labelled with an **F** prefix and the rotational ones with an **M** prefix. Here only hinged supports will be defined, since the design part of STAAD.Pro will not even recognise them as discussed in Section 3.5.1. One support is created where **FX**, **MY** and **MZ** are checked and another where only **MY** and **MZ** are checked. These two supports are then assigned to the nodes, one on each end.

The next subpage is the **Load & Definition** one, where a load is defined by selecting **New** on the data area on the right hand side. In the **Create New Definitions / Load Cases / Load Items** window that appears, a new load case must first be added under the **Load Case** tab and **Primary**, where the **Loading Type** can be set to **Live**. When the load case has been added, the load is defined under the **Load Items** tab and **Member Load**. A concentrated load is defined by adding a **Concentrated Force** with any negative magnitude **P** acting at a distance from beams end **d1** equal to half of the beam length, and distance from shear centre **d2** as zero, in the **GY** direction (see Figure 4.17). In the same way a distributed load over the whole span acting at the shear centre can be defined as **Uniform Force** with any negative magnitude **W1**, the distances **d1**, **d2** and **d3** as zero and in the direction **GY** (see Figure 4.18). The load defined must then be assigned to the beam.

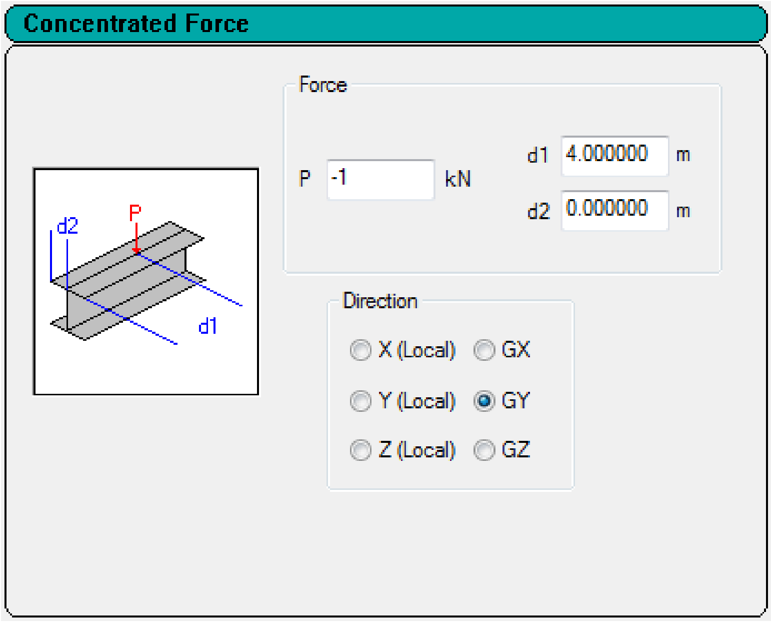



*Figure 4.17* *Definition of a concentrated load in STAAD.Pro acting downwards at the shear centre at mid-span for an 8 meter long beam. The diagram on the left can be misleading, since the force* ***P*** *must be assigned a negative value to actually act downwards and the distance* ***d2*** *is measured perpendicularly from the shear centre to the plane of loading.*

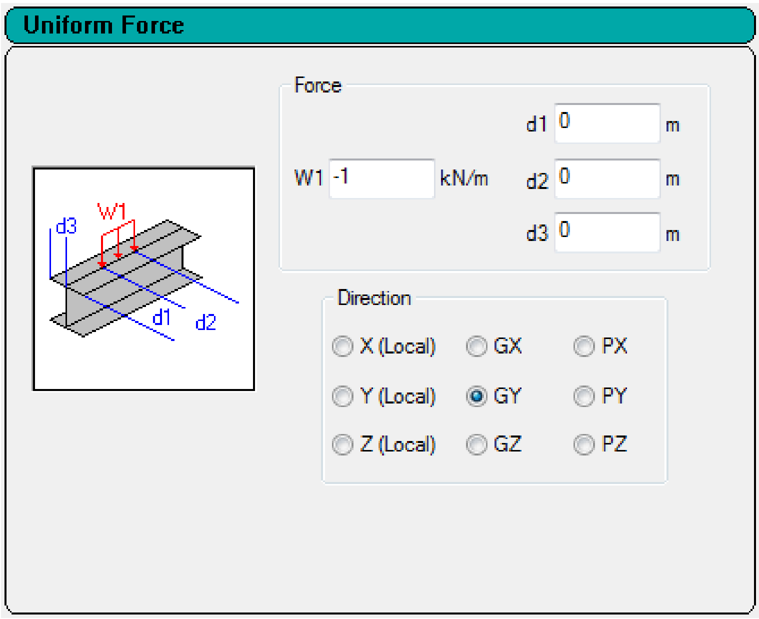



*Figure 4.18* *Definition of a uniformly distributed load in STAAD.Pro acting downwards at the shear centre over the entire span of the beam. The diagram on the left can be misleading, since the force* ***W1*** *must be assigned a negative value to actually act downwards and the distance* ***d3*** *is measured perpendicularly from the shear centre to the plane of loading.*

To finalise the modelling part an analysis type is specified by choosing the **Analysis/Print** page and the **Perform Analysis** tab with the **Print Option** specified as **All**. Then the design page is opened and code checks are made, as described in Section 3.5.1, where most of the design parameters are redefined for the analytical evaluation of $M_{cr}$, with the corresponding values. The C-factors are taken from Table 2.1. Different degree of end restraint is modelled by manually specifying the value for the unrestrained length of the beam. The vertical position of the PLA is varied by manually changing the $z_g$ parameter. The analysis is run by expanding the **Analyze** tab and clicking **Run Analysis**, where the option of opening the output file is offered subsequently.

### 4.2.5 Modelling with beam elements in ADINA

When modelling a beam which is only intended to be loaded in the vertical plane through its shear centre, like the monosymmetric I-beam described in Section 4.1.1, it is considered applicable to use warping beam elements and perform a linearized buckling analysis in ADINA. The procedure of modelling such a beam in that way is quite effective and can even be implemented in a design process.

Models in ADINA can be saved as two different types of files; ADINA-IN Database files that include everything that has been done with the model, from defining points to rotating the view for example, and ADINA-IN Command files that are more raw and only include necessary modelling commands to retrieve the model once reopened. When modelling in ADINA the units don't have to be chosen, but it is important to use consistent set of units such as length in meters, force in newtons, mass in kilograms and time in seconds (ADINA 2013, p. 2).

Once the software has been launched, choosing the structural engineering part is made by selecting **ADINA Structures** from the program module drop-down list, which is located in the bottommost toolbar as can be seen in Figure 3.1. The theory for large deformations is needed, as discussed in Section 2.2.1, which is selected by expanding the **Control** menu, **Analysis Assumptions**, opening **Kinematics** and setting **Displacement/Rotations** to **Large**, as is showed in Figure 4.19.

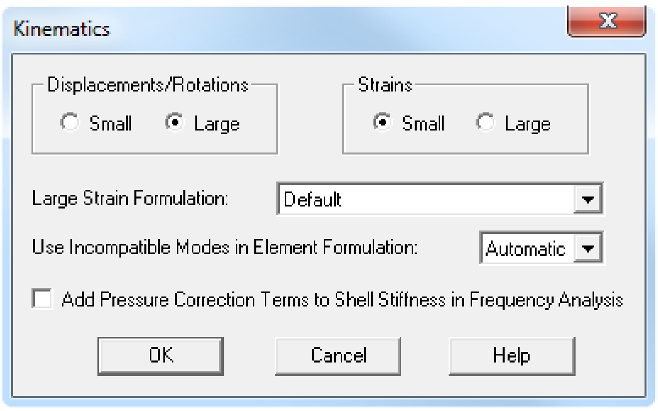



*Figure 4.19 The kinematic formulation window in ADINA, where the theory for large deformations is chosen.*

The modelling procedure will be done in the following order:

- Geometry
- Materials & sections
- Element groups
- Element subdivision
- Rigid links
- Support conditions
- Loads
- Analysis

For a beam model that will include mid-span concentrated load, tree points have to be created; one at each end of the beam and one at the centre. Points are defined by expanding the **Geometry** menu and opening **Points**, or by the **Define points** icon on the toolbar. The three points can be defined for an 8 meter long beam as shown in Figure 4.20, where point 1 has the coordinates (0,0,0), point 2 (8,0,0) and point 3 (4,0,0). In that way the x-axis will be the longitudinal axis of the beam.

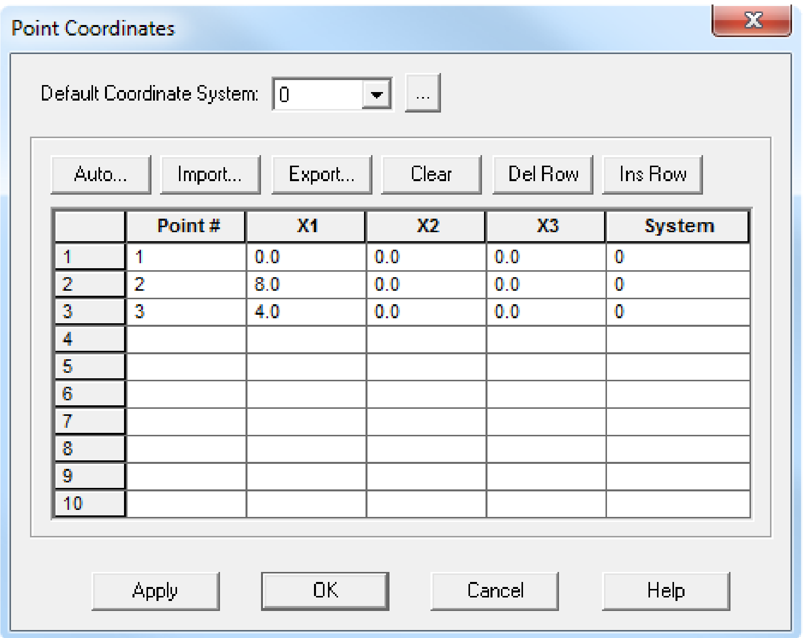



*Figure 4.20 Definition of geometry points in ADINA.*

Two lines have to be created, connecting the three points. This is done by expanding **Geometry**, **Lines** and opening **Define** or by the **Define Lines** icon in the toolbar. In the **Define Line** window click **Add** to add the first line, make sure the **Type** is **Straight** and specify its **Point 1** and **Point 2** as 1 and 3 respectively, as shown in Figure 4.21. In the same way the second line is created by clicking the **Add** button again and specifying **Point 1** and **Point 2** as 3 and 2 respectively.

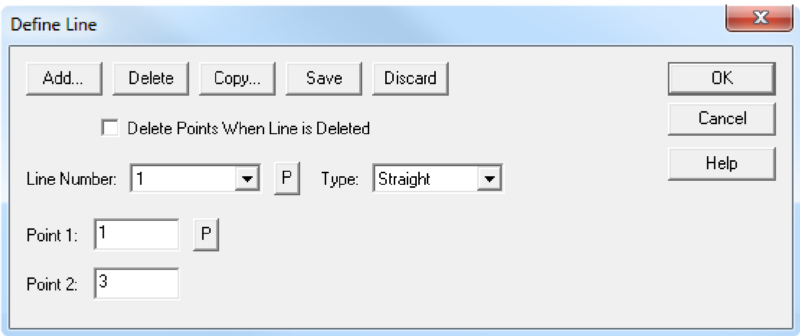



*Figure 4.21 Definition of geometry lines in ADINA.*

The material is defined by expanding the **Model** menu, **Materials** and open **Manage Materials** or by using the corresponding icon in the toolbar. The desired linearly elastic material is chosen by clicking **Isotropic** under **Elastic**. In the **Define Isotropic Linear Elastic Material** window **Add** is clicked to add the material. The Young's modulus and Poisson's ratio are specified, as shown in Figure 4.22. The rest of the input parameters do not need to be specified for the intended analysis.

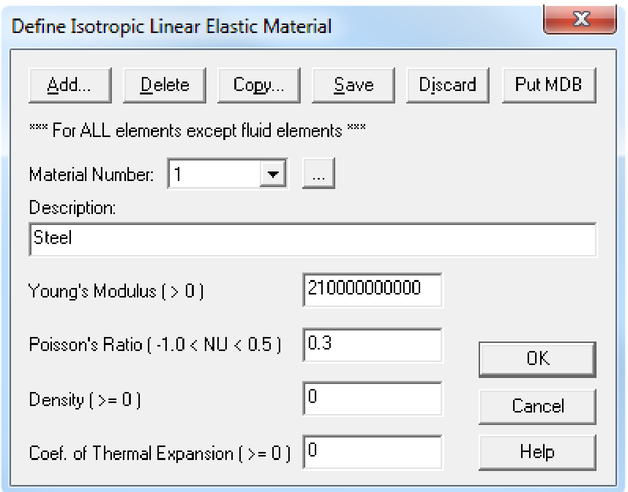



*Figure 4.22 Definition of a linearly elastic material in ADINA. Here the **Description** has been set as **Steel**.*

The cross section can be created by expanding the **Model** menu and opening **Cross-Sections**, or with the corresponding icon in the toolbar, and clicking **Add** in the **Define Cross Section** window. Two types of cross sections can be used to model the monosymmetric I-beam; a cross section of the type **I-Beam** where only the dimensions need to be specified and the sectional parameters will be calculated by the program or a cross section of the type **General** where all sectional parameters need to be specified manually. If the monosymmetric I-section is created with the **I-Beam** type, its dimensions are specified and the corresponding sectional parameters will be calculated by the software using equations (3.11) to (3.21).

Here the monosymmetric I-section is created with the **General** type, which gives the opportunity to manually specify the values used for the sectional parameters, to give a better comparison between software. The **Define Cross Section** window using a **General** type of cross section and the corresponding sectional parameters for the monosymmetric I-section, described in Section 4.1.1, can be seen in Figure 4.23. Equations (3.17) to (3.21) are used to calculate the Wagner effect constants.

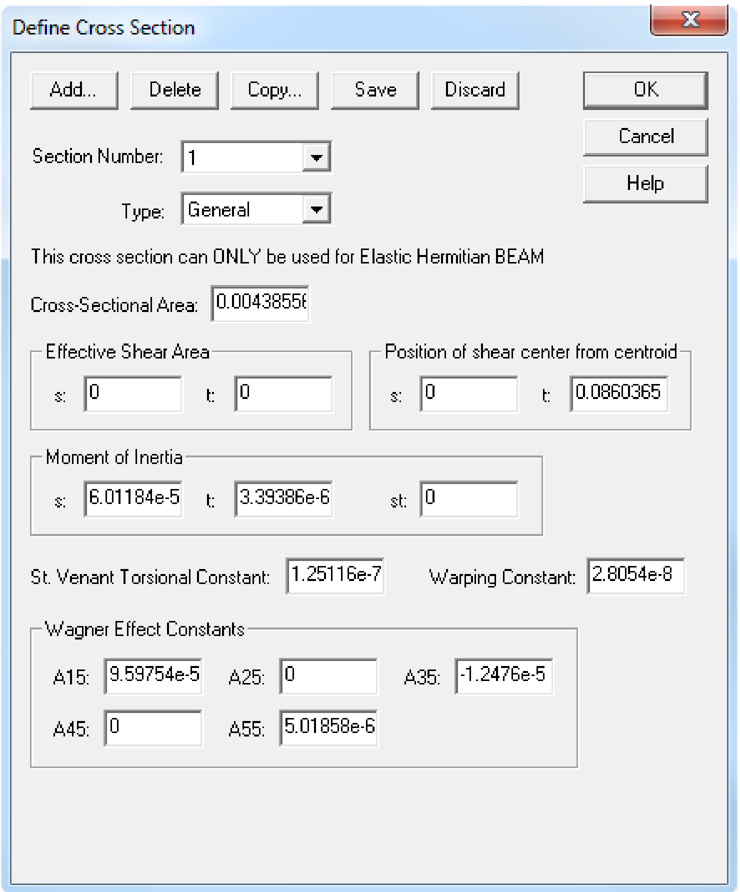



*Figure 4.23 Definition of a cross section of the type **General** in ADINA. The inserted parameters correspond to the monosymmetric I-section described in Section 4.1.1.*

To define the finite elements a so called element group is defined. By expanding the **Meshing** menu and selecting **Element Groups**, or by clicking on the corresponding icon in the toolbar, the **Define Element Group** window is opened. A new element group is created by clicking on **Add** and its **Type** is set to **Beam**. Warping is included, by checking the **Include Warping DOF** option, and **Displacements** can either be set to **Large** or **Default**, since the theory for large deformations has already been set as default. Under **Stiffness Definition** the **Use Material and Cross Section** option is used and the previously defined material and section chosen. The rest is left as default, as can be seen in Figure 4.24.

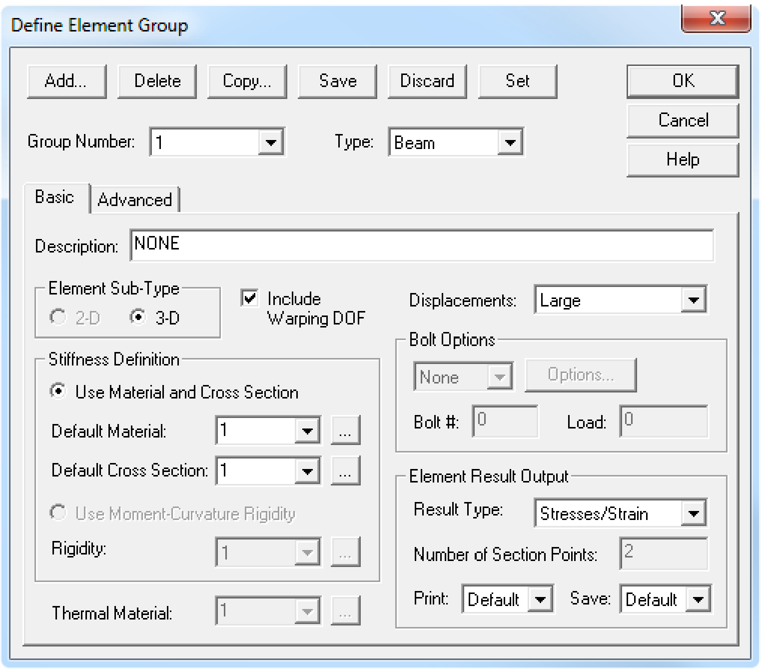



*Figure 4.24 Definition of an element group using warping beam elements in ADINA.*

The finite elements are assigned to the geometry by subdividing the geometry and creating a mesh. The subdivision is made by defining the mesh density, which can be accessed through the **Meshing** menu, **Mesh Density** and **Line** or the corresponding icon in the toolbar. In the **Define Line Mesh Density** window the two lines are specified by selecting line 1 in the **Line Number** field and adding also line 2 in the **Also Assign to Following Lines** list. **Use Number of Divisions** is chosen as the **Method** and **Number of Subdivisions** is set to 50, as shown in Figure 4.25. That means the beam in total will be made out of 100 elements of equal size.

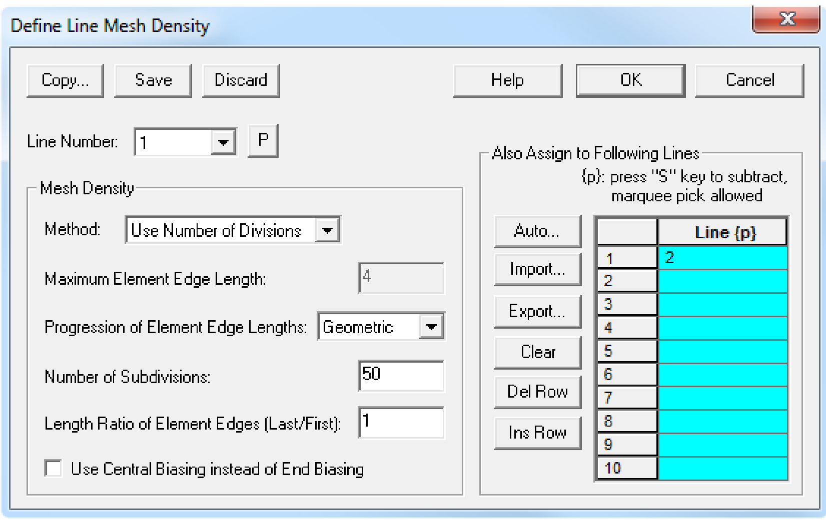



*Figure 4.25 Definition of a line mesh density in ADINA.*

The mesh is created through the **Meshing** menu, **Create Mesh** and **Line** or the corresponding icon in the toolbar. In the **Mesh Lines** window the **Type** is set to **Beam**, the created element group is chosen and the **Orientation Vector** is defined as (0,1,0) so the cross section will be orientated as defined in this thesis. In the **Lines to be Meshed** list the two lines are inserted, as shown in Figure 4.26.

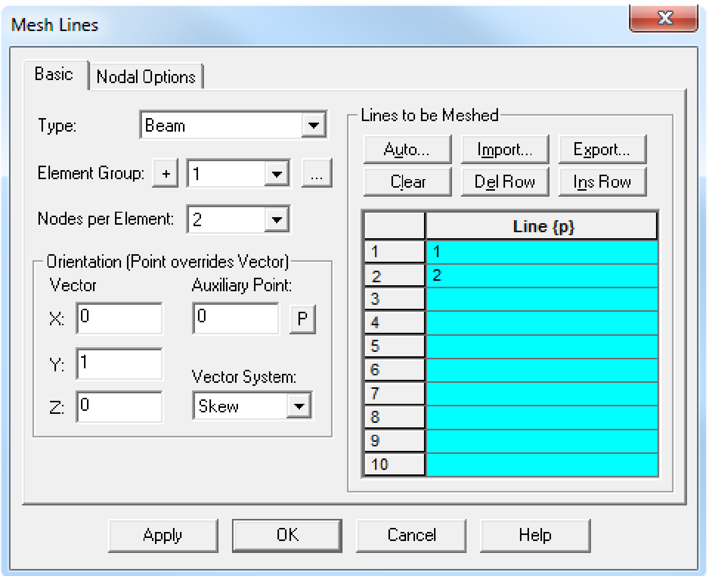



*Figure 4.26 A line mesh created in ADINA.*

A special constraint called rigid link can be used to connect two nodes; a master node and a slave node, so when the nodes displace the slave node translates the same distance as the master node and the slave node rotates in the same way as the master node (ADINA 2013, p. 837). When modelling with beam elements the beams defining line in ADINA lies at the centroid of the cross section. A rigid link is used for the monosymmetric I-beam when the vertical position of the PLA is varied. Should a point load be acting at the top of the cross section a point needs to be created, as described above, with the corresponding coordinates. Then through the **Model** menu, **Constraints** and **Rigid Links** a rigid link is added with point 3 as a slave point and the point of load application as a master point with all six translational and rotational degrees of freedom as **Slave Constrained DOF**, as shown in Figure 4.27.

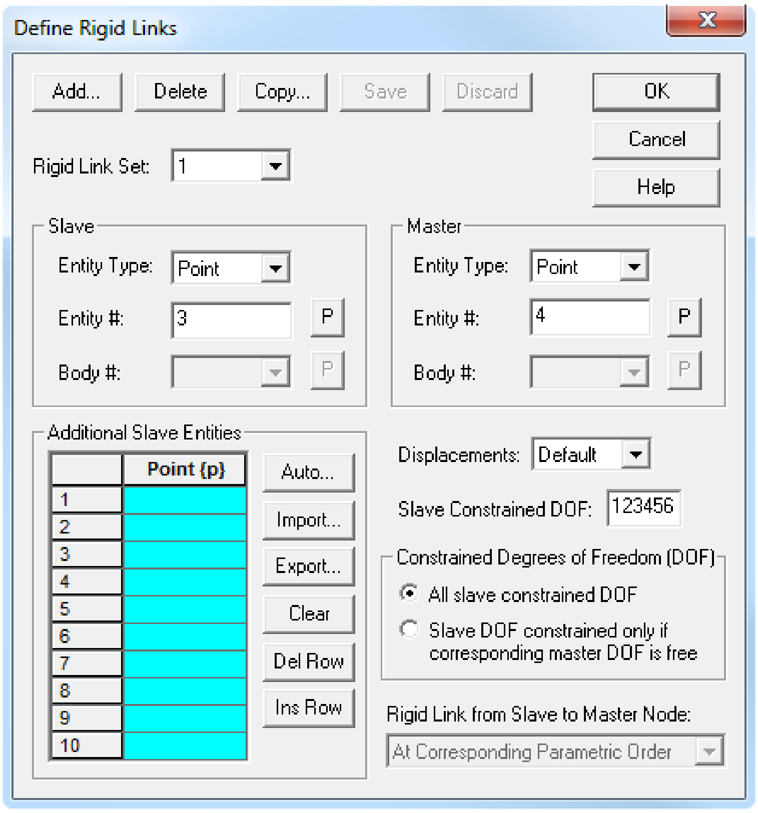



*Figure 4.27 Definition of a rigid link between two points in ADINA, where all possible degrees of freedom are constrained.*

Support conditions are created by defining fixities through the **Model** menu, **Boundary Conditions** and **Define Fixity**. Using hinged supports, see Section 2.1.8 and especially Figure 2.6, two fixities are defined, one for each end. One has x-translation fixed but it is kept free in the other one. In the **Define Fixity** window a fixity is created by clicking **Add** and specifying a name. For one end the following is chosen as **Fixed Degrees of Freedom**; translations in x-, y- and z- directions and rotation in x-direction, as shown in Figure 4.28. For the other end another fixity is added with y-, and z- translations and x-rotation fixed. Using these support conditions will result in the effective length factors $k_z = k_w = 1$ for the beam. For modelling another degree of end restraint for z-rotation and warping the corresponding degrees of freedom, noted as **Z-rotation** and **Beam Warp**, need to be added to **Fixed Degrees of Freedom** in the appropriate manner. To obtain $k_z = k_w = 0.7$ the z-rotation and warping degrees of freedom are fixed at one end, but free at the other, and in the case of $k_z = k_w = 0.5$ both degrees of freedom are fixed at both ends.

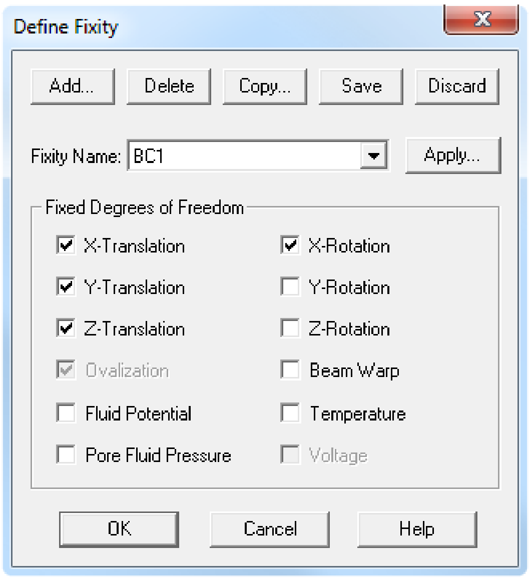



*Figure 4.28 Example of a definition of a hinged support condition in ADINA.*

The support conditions are assigned to the structure by expanding the **Model** menu, **Boundary Conditions** and opening **Apply Fixity** or with the corresponding icon in the toolbar. One of the previously defined fixities is applied to point 1 and the other to point 2, as shown in Figure 4.29.

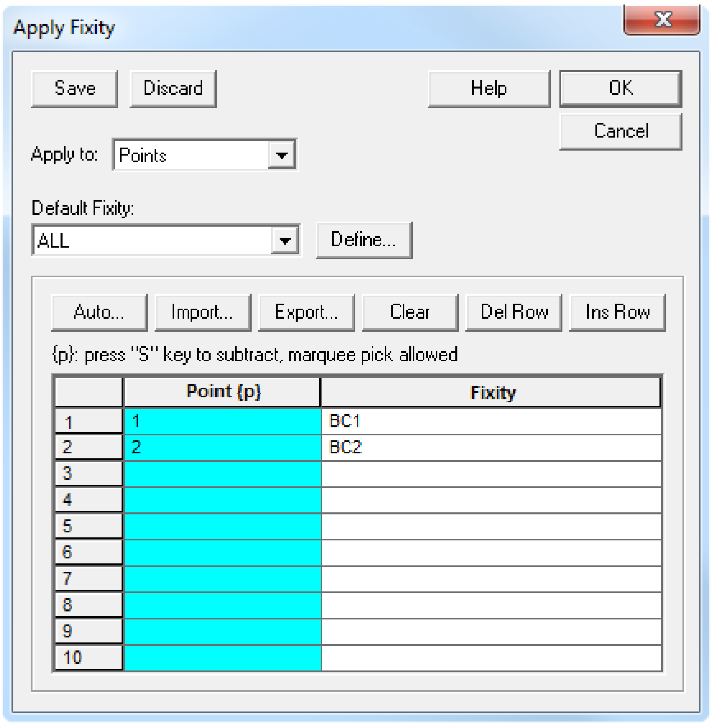



*Figure 4.29 Support conditions assigned to points in ADINA.*

Loads are defined through the **Model** menu, **Loading** and **Apply** or the corresponding icon in the toolbar. When creating a concentrated load, the **Load Type** in the **Apply**

**Load** window is set to **Force** and the **Define** button is clicked. By clicking the **Add** button in the **Define Concentrated Force** window, specifying the **Magnitude** as 10000 and the **Force Direction** as -1 in the **Z** field, a 10 kN concentrated force is defined and acting vertically downwards, as shown in Figure 4.30. The load is then applied in the **Apply Load** window by choosing the **Load Number** of the previously defined load and specifying the **Point** as 3 to have the load acting at mid-span. A uniformly distributed load is created in a similar way, only then the **Load Type** is set to **Distributed Line Load** and it is only defined by a negative **Magnitude** in the **Define Distributed Line Load** window should it be acting downwards. Then it is applied by specifying the line it acts on and the **Load Direction** in the **Apply Load** window.

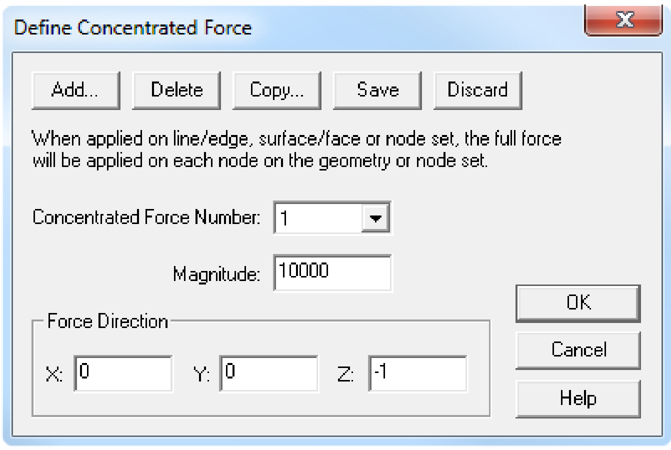



*Figure 4.30 Definition of a* $10\,kN$ *point load acting vertically downwards in ADINA.*

The type of analysis to be performed is selected from the **Analysis Type** drop down list, below the toolbars, as **Linearized Buckling**. If the **Analysis Options** button next to the drop down list is clicked, various options can be specified such as number of buckling modes to be found and which **Buckling Analysis Method** should be used. As discussed in Section 3.1.1.1, the classical formulation is more accurate and should therefore be used as the **Buckling Analysis Method**.

The analysis is performed by expanding the **Solution** menu and selecting the **Data File/Run** option, or by the corresponding icon in the toolbar. In the **Create the ADINA Data File** window that appears, the file is given a name and saved. Then the analysis will be executed and its status can be overseen in a window that appears and is shown in Figure 4.31. From there the output file can be opened and if the solution is successful the resulting critical buckling load factor $\lambda_1$, described in Section 3.1.1.1, can be viewed under **Linearized Buckling Load** as shown in Figure 4.32.

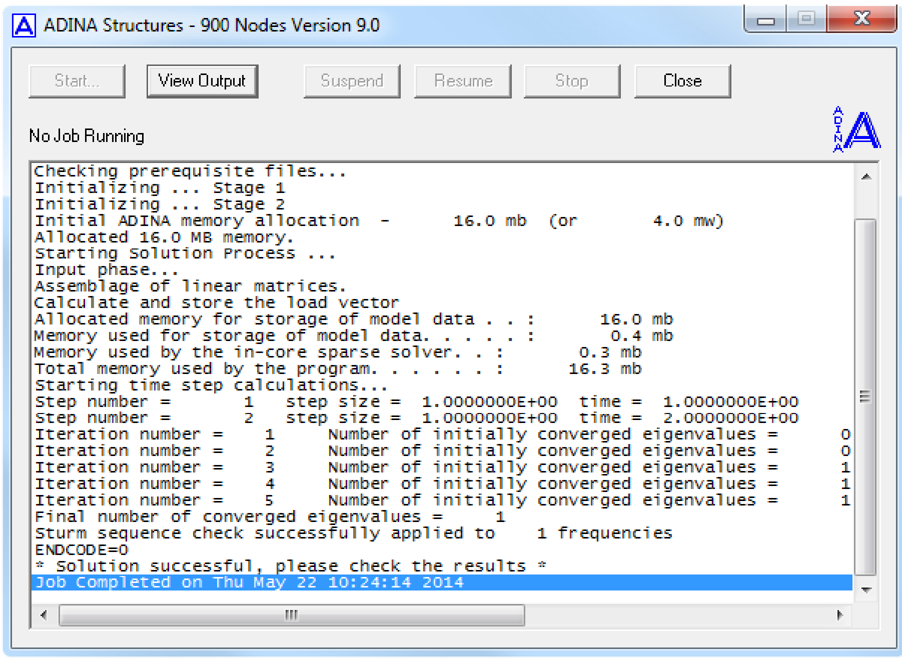



*Figure 4.31 Execution of a linearized buckling analysis in ADINA.*

```
 L I N E A R I Z E D   B U C K L I N G   L O A D


      BUCKLING LOAD NO.   1
      CRITICAL LOAD FACTOR (LAMBDA) EQUALS    0.317311E+01
1PROGRAM ADINA - VERSION 9.0.1 (build 01.30.2014)     *** NO HEADING DEFINED ***
```

*Figure 4.32 Example of a part of an output file resulting from a linearized buckling analysis in ADINA, where the critical buckling load factor can be viewed.*

### 4.2.6 Modelling with shell elements in ADINA

Compared to the monosymmetric I-beam, a beam with a channel section is more complex to model, since both its centroid and shear centre lie outside its actual cross section. Using beam elements to model such a beam would result in complications, when for example support conditions or loads are to be applied at the sections shear centre, or anywhere else except at the centroid. When modelling a beam with a channel section it would be more applicable to utilise shell elements and either perform a linearized buckling analysis or a collapse analysis with the LDC method.

The modified version of the UPE160 channel beam, described in Section 4.1.2, will be modelled using shell elements in ADINA. The shell elements used are four node 2D shells that are based on the Timoshenko beam theory and the Reissner/Mindlin plate theory (ADINA 2013, p. 176). The Newton-Cotes integration will be used with 7 integration points through the shell thickness and 4 in the plane of the shell. The reason for the choice of the integration method is that Newton-Cotes integration gives results on the surface of the shell but Gauss integration doesn't, as is illustrated in Figure 4.33 (ADINA 2013, pp. 196-197).

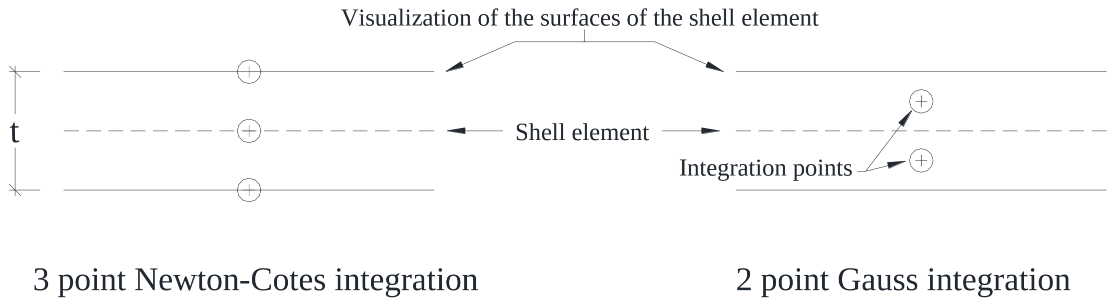



*Figure 4.33 Illustrative comparison of the Newton-Cotes and Gauss integration methods in ADINA through the thickness t of shell elements.*

In order to make the shell model, surfaces are defined by locating their corner points (see Figure 4.34). Those surfaces define where the shell elements will be located. Additional points are created in the end of the beam and at mid-span, located at the gravity centre, see points 13-15 in Figure 4.34, and at the shear centre of the section, see points 16-18 in Figure 4.34. These additional points offer the opportunity for application of the support conditions and loads.

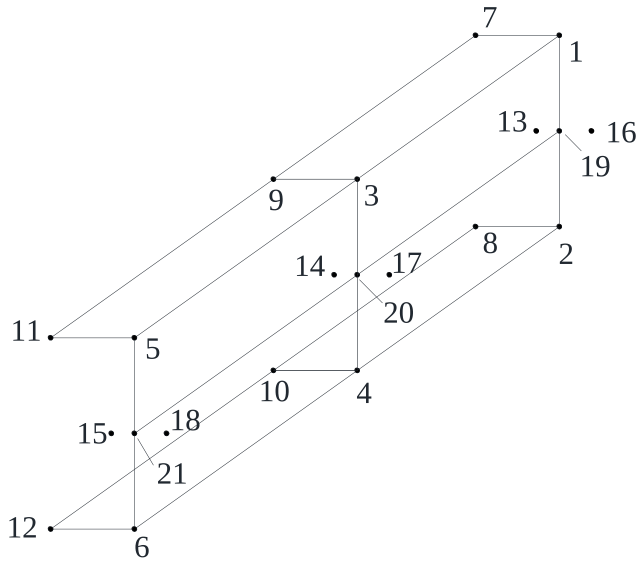



*Figure 4.34 Illustration of a beam with channel section modelled with shell elements in ADINA, where numbering of nodes is noted. This model is subdivided at the centre of the span to provide points for load application.*

The flanges of the channel beam are modelled to the centreline of the web. The same applies to the web; it is modelled between the centrelines of the flanges. This means that two squares in the cross section, at the ends of the web, do not have any material defined. However, a square with the same size has the material defined twice (see Figure 4.35).

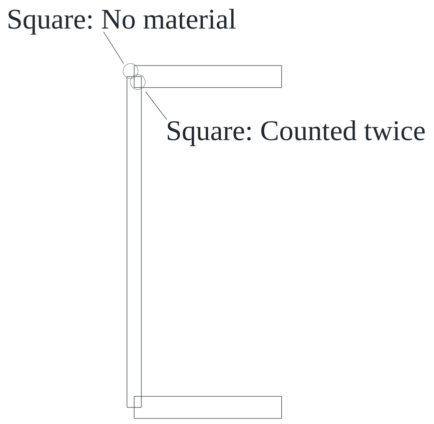



*Figure 4.35 Illustration of a channel cross section modelled with shell elements in ADINA and especially the intersection of cross section parts.*

The modelling process of a channel beam in ADINA will not be described in as much detail as for the monosymmetric I-beam in Section 4.2.5, except how the shell elements are implemented. Examples of ADINA-IN Command files for both types of loading and for both centrically loaded and eccentrically loaded beams can be found in Appendix B, in Sections B.2 and B.3, respectively.

The kinematic formulation is specified in the same way as for the model with beam elements, to make use of the theory for large deformations, and the points described in Figure 4.34 are created in the same way as with the beam elements. Instead of creating lines to connect the points, surfaces are defined by expanding the **Geometry** menu, **Surfaces** and opening **Define**, or by the corresponding icon in the toolbar. The surfaces are added, the **Type** is set to **Vertex** and the four corner points of each surface are specified. An example of a surface definition can be seen in Figure 4.36

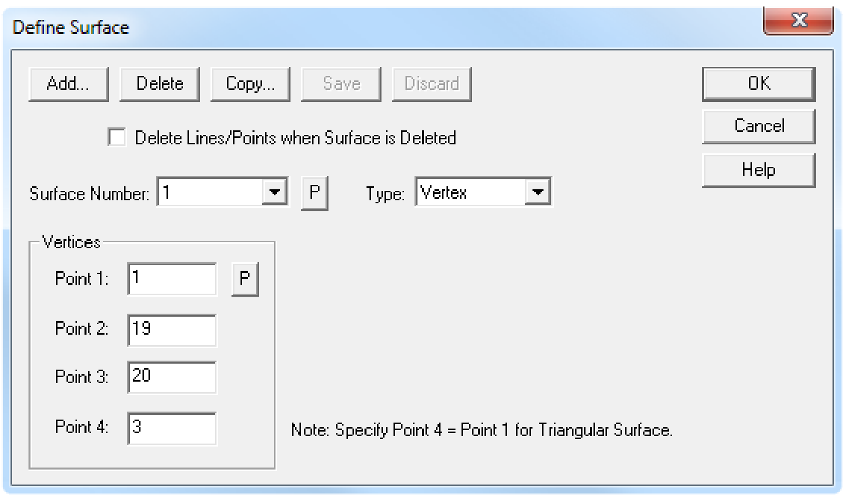



*Figure 4.36 Definition of a surface in ADINA.*

The material is linearly elastic as before, but a different kind of element group is used. Since the thickness of the web and the flanges is different for the channel section, two element groups are created, one corresponding to each thickness. In the **Define Element Group** window, see Figure 4.24, the **Type** is set to **Shell** and under the **Basic** tab the material is chosen, the thickness specified in each case as the **Default Element Thickness**, the **Newton-Cotes** specified as **Integration Type** and the **Integration Order** set to 7 through the shell thickness. Under the **Advanced** tab in the **Define Element Group** window the number of integration points in the plane of the shell is chosen by selecting 2 in both directions for the **Number Integration Order (Membrane)**.

The element subdivision is performed in a similar way as for the beam elements, i.e. by defining the mesh density for the different surfaces through the **Meshing** menu, **Mesh Density** and **Surface**. The student version of ADINA used in this thesis only allows for 900 nodes to be created, which affects the choice of the mesh density A convergence study has been performed, and a comparison of mesh densities is presented in Figure 4.37. The following mesh density is chosen:

- 4 elements over the width of each flange,
- 6 elements over the height of the web,
- 10 cm long elements in the longitudinal direction of the beam.

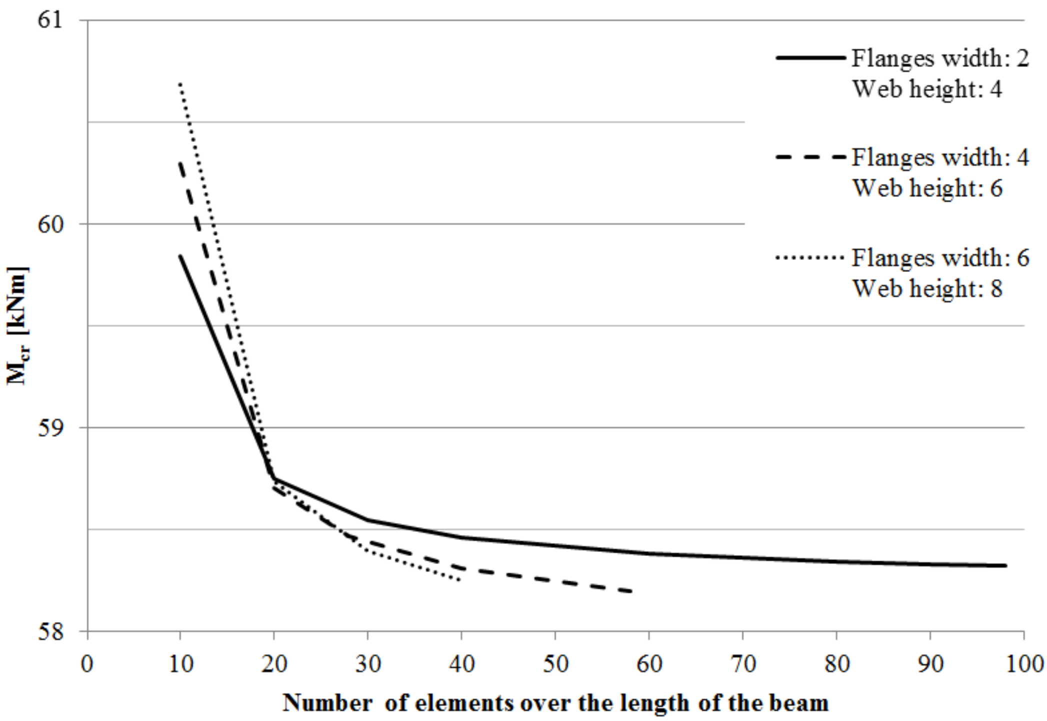



*Figure 4.37 Comparison of various mesh densities for the* 2.8 *m long channel beam subjected to a concentrated mid-span load, obtained with a linearized buckling analysis in ADINA. In the chosen mesh density, 4 elements are over the width of the flanges, 6 elements over the height of the web and* 10 *cm long elements over the length of the beam (28 elements for this beam). For the chosen mesh density, similar results for $M_{cr}$ are obtained as if a denser mesh is used.*

The number of elements in the longitudinal direction of the beam is therefore determined by the length of the beam. This choice of subdivision allows for beams up to 5.8 m long to be modelled, without exceeding the limit for the number of nodes, and still gives similar results for the shorter beams as if a denser mesh is used. In Figure 4.38 an example of the **Define Surface Mesh Density** window for when the surfaces representing the beam flanges are subdivided.

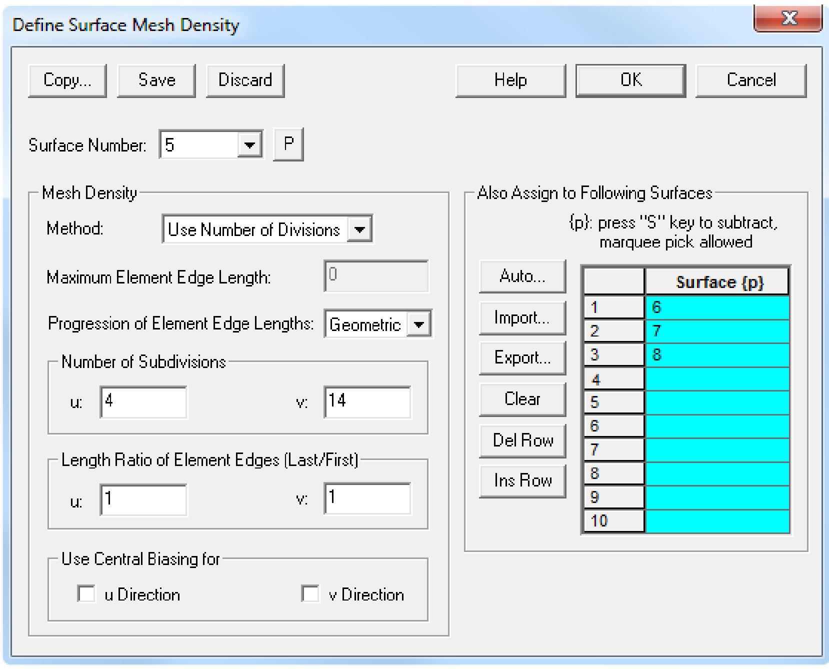



*Figure 4.38 Definition of a surface mesh density in ADINA.*

Creating a mesh for a surface, and by that connecting the element groups to the elements, is similar to creating a mesh for a line. In the **Mesh Surfaces** window the **Type** is **Shell**, **Nodes per Element** is specified as 4, the **Preferred Cell Shape** is set as **Quadrilateral** and the surfaces are listed that represent the element group chosen. A point mesh is created for the points that lie outside the cross section and will be used for support conditions or loading, so they will be assigned nodes. The point mesh is created similarly to other meshes, and only the point numbers are listed.

The use of rigid links is important for the shell model of the channel beam, since the support conditions are to be applied at the shear centre of the section and so are the loads in some cases. By constraining the appropriate degrees of freedom, the behaviour of the beam model when loaded will be comparable to if the shear centre is an actual part of the cross section. In order to allow warping of the beam, two separate rigid links are made for each end of the beam, to connect the supported shear centre to the rest of the cross section, as illustrated in Figure 4.39. First all the degrees of freedom of the nodes in the end sections web are constrained to the degrees of freedom of the shear centre, by creating a rigid link with the lines along the webs height as slave and the point of shear centre as master (see Figure 4.27 for creating a rigid link). Then the y- and z-translations of the nodes in the end sections flanges are constrained to the corresponding degrees of freedom of the shear centre, by creating a rigid link with the lines along the flanges width as slaves, the shear centre as master and **Slave Constrained DOF** specified as **23**. In that way the end of the flanges are able to displace in the x-direction as well as rotate relative to the shear centre, which makes warping possible.

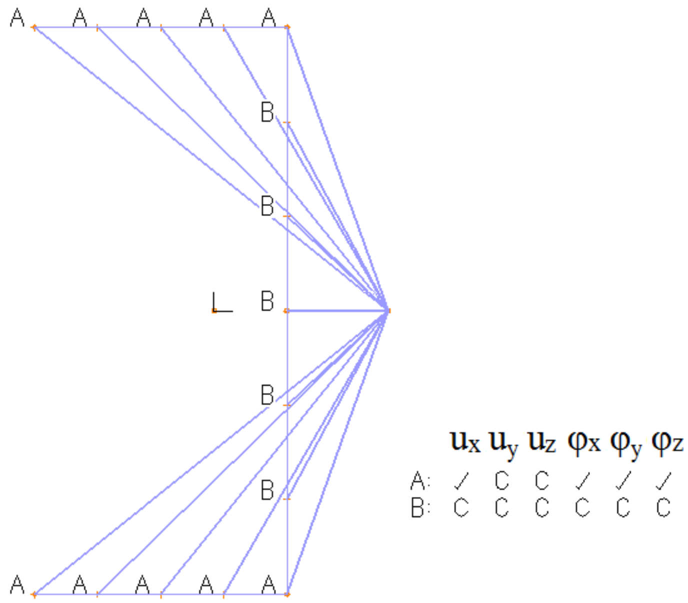



*Figure 4.39 Illustration of rigid links for an end section of a beam in ADINA, where support conditions are applied. The y-translation $u_y$ and the z-translation $u_z$ of all the nodes in the flanges of the end section, noted with A, are constrained to the corresponding DOFs of the shear centre. All the six DOFs of the nodes in the web of the end section, noted with B, are constrained to the corresponding DOFs of the shear centre. In the legend, constrained DOFs are noted with C.*

When applying a concentrated load to the shell element model, no matter if the point of load application is actually a part of the cross section or not, a rigid link is used to constrain the webs nodes in that section to the PLA. In that way distortion of the cross section where the load is applied is avoided. Thus it is assumed that the section of load application remains intact, even though by doing so the section becomes somewhat stiffer than it else would be. The rigid link is created by specifying the lines along the height of the web in the section of load application as slaves, the PLA as master and selecting all the degrees of freedom as **Slave Constrained DOF**. In Figure 4.40 an example of a channel beam subjected to concentrated loading can be seen, where a rigid link constrains the point of load application to the beam.

When applying a uniformly distributed load to the shell element model, a line of load application is defined if the load is to be applied outside the cross section, for example along the shear centre line of the beam. This auxiliary line is modelled with beam elements in order to be a part of the FEM model. The beam elements are created and assigned to the line in the same way as described in Section 4.2.5. The stiffness of the beam elements is set as low as possible, so the influence of the auxiliary line on the stiffness of the channel beam is minimized. A rigid link is then created with the line of load application as slave and the line along the centre of the channels web as master, in order to connect all the degrees of freedom of the two lines together. In Figure 4.41

an example of a channel beam subjected to a distributed load can be seen, where a rigid link constrains the line of load application to the beam.

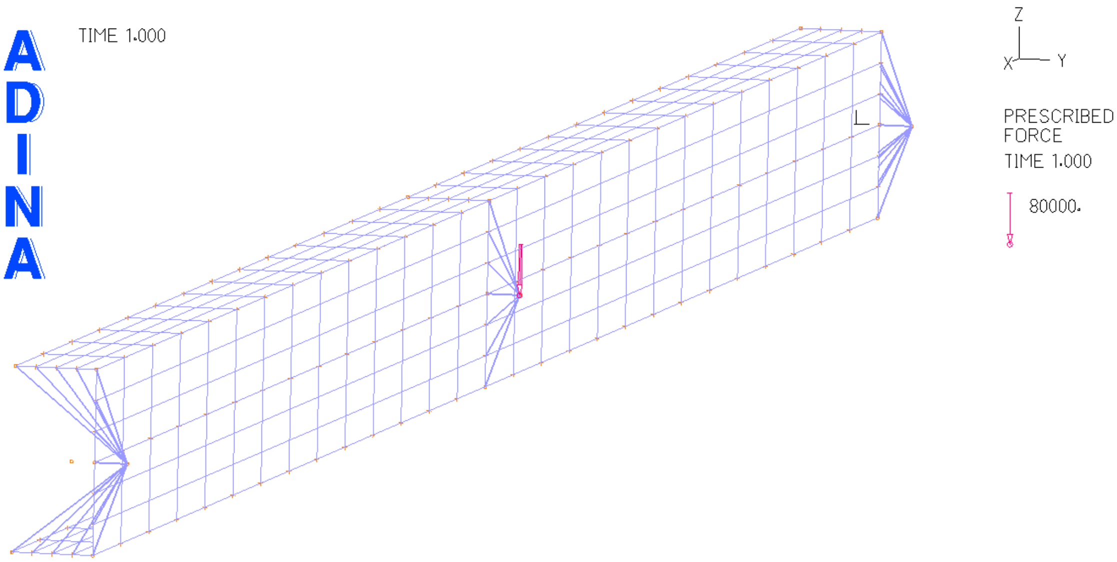



*Figure 4.40* *A shell element model of a channel beam subjected to a concentrated mid-span load in ADINA. The constraining of the degrees of freedom at the beams ends to the supported nodes can be seen, as well as the connection between the point of load application and the channel beam.*

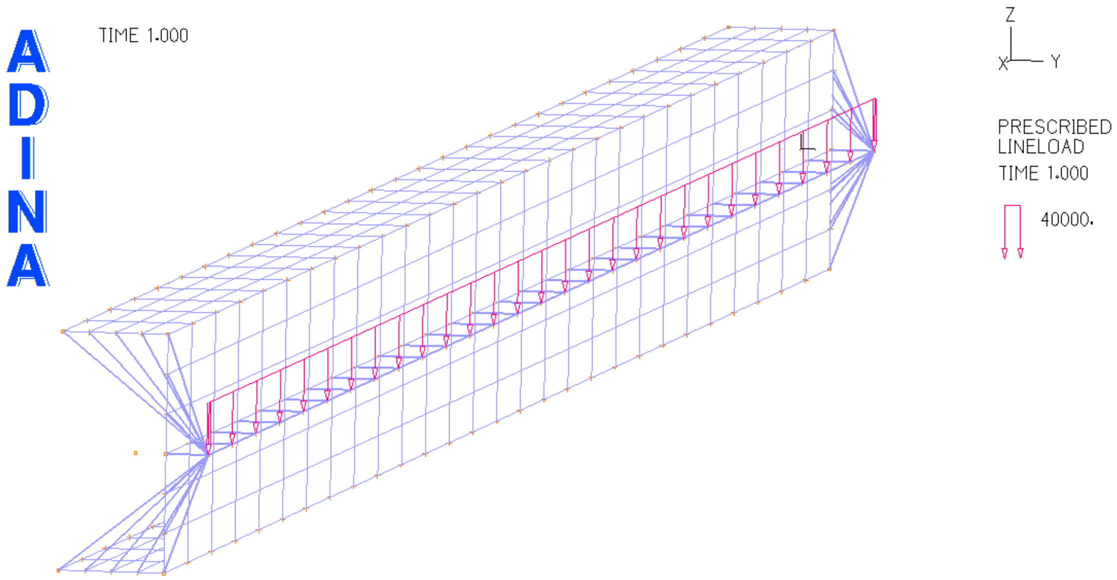



*Figure 4.41* *A shell element model of a channel beam subjected to a uniformly distributed load over the whole span in ADINA. The constraining of the degrees of freedom at the beams ends to the supported nodes can be seen, as well as the connection between the line of load application and the channel beam.*

For the model of the channel beam made of shell elements, both linearized buckling analysis and collapse analysis with the LDC method will be performed in order to estimate $M_{cr}$. With linearized buckling analysis an approximate solution of the critical load can be achieved, but can only be employed for the simplest cases of bifurcation buckling (Earls 2007). Therefore the focus will be set on performing a collapse analysis with the LDC method, which gives a more accurate solution for the first critical load (R. Wang, personal correspondence, March 19, 2014).

Linearized buckling analysis is performed in the same way when using shell elements as when using beam elements, which can be seen in Section 4.2.5. A collapse analysis is specified in the same way as linearized buckling analysis, only by choosing **Collapse Analysis** from the dropdown list. In the analysis option an initial displacement in the direction of the load must be specified for a point or a node where load is applied. A point or a node at the beams mid-span is chosen, the **Degree of Freedom** is specified as **Z-Translation** and a very small negative **Displacement** is sufficient. The **Maximum Allowed Displacement** is specified so that the analysis will terminate before a very unreasonable displacement is achieved. It is convenient to check the **Continue after First Critical Point is Reached** option to obtain a full load-displacement relation around the critical point.

For both types of analyses, the iteration tolerance needs to be tightened, using the energy and force/moment convergence criterion mentioned in Section 3.1.1.2. This is specified by expanding **Control** and opening **Solution Process** and then **Iteration Tolerances**. The **Convergence Criteria** is set as **Energy and Force**, the **Energy Tolerance** can be set to $1 \cdot 10^{-6}$ or even lower and the **Force (Moment) Tolerance** to $1 \cdot 10^{-3}$ (R. Wang, personal correspondence, March 18, 2014).

For the incremental collapse analysis the number of time steps should be defined. This is done by expanding the **Control** menu, opening **Time Step** and specifying **Number of Steps** as 100, for example, in the first row in the **Define Time Step** window. Introducing initial geometric imperfection of the first buckling mode shape acts as a small disturbance of the beams equilibrium, as described in Section 2.2.1, and thus helps initiating lateral-torsional buckling. The amplitude of the imperfection should be very small, since $M_{cr}$ is in theory the buckling moment capacity of an ideal beam, i.e. of an imperfection free beam. The initial imperfection is defined by expanding the **Model** menu, **Initial Conditions** and opening **Imperfection**. In the **Define Imperfection** window either a point or a node at the beams mid-span is chosen when the first buckling mode shape is used. The **Buckling Mode** is then set to 1, the number of the point or the node specified, the **Direction** is set as **Y-Translation** and the **Displacement** a small value, for example 0.0001 (0.1 mm). When running the collapse analysis, ADINA asks for a mode shape file to create the imperfection with the amplitude specified. The mode shape file chosen is the resulting file from a linearized buckling analysis of the same case.

Performing a collapse analysis successfully will result in the creation of an ADINA Porthole file, which contains the results from the analysis. Opening the file in ADINA gives the opportunity to perform post-processing of the results. Most relevant for the current study, is to visualize the deformed shape of the beam at different time steps and access numerical data for different variables, in order to obtain the load-displacement relation.

### 4.2.7 Comparison of sectional parameters in different software

This section contains a numerical comparison of the values for sectional parameters for the two sections studied, using the different software introduced before. The comparison can be seen in Table 4.4 for the monosymmetric I-section, described in Section 4.1.1, and in Table 4.5 for the channel section, described in Section 4.1.2.

*Table 4.4 Numerical comparison of sectional parameters for the monosymmetric I-section described in Section 4.1.1, using the different calculation methods listed in Section 4.1.1.*

| Parameter | ADINA | Colbeam | LTBeam | SAP2000 | STAAD.Pro | 3-factor formula |
|---|---|---|---|---|---|---|
| $I_t$ $[\cdot 10^5 \text{ mm}^4]$ | 1.251 | 1.251 | 1.216 | 1.191 | 1.251 | 1.251 |
| $I_w$ $[\cdot 10^{10} \text{mm}^6]$ | 2.805 | 2.799 | 2.853 | N.A.3) | 2.805 | 2.805 |
| $I_y$ $[\cdot 10^7 \text{ mm}^4]$ | 6.012 | 6.012 | 6.012 | 6.012 | 6.012 | 6.012 |
| $I_z$ $[\cdot 10^6 \text{ mm}^4]$ | 3.394 | 3.394 | 3.394 | 3.394 | 3.394 | 3.394 |
| $z_j$ [mm] | 103.77 | 90.00 1) | 103.49 4) | N.A.2) | N.A.2) | 103.77 |

*Table 4.5 Numerical comparison of sectional parameters for the channel section described in Section 4.1.2, using the different calculation methods listed in Section 4.1.2.*

| Parameter | ADINA | Colbeam | LTBeam | SAP2000 | 3-factor formula |
|---|---|---|---|---|---|
| $I_t$ $[\cdot 10^4 \text{mm}^4]$ | N.A2) | 5.380 | 5.823 | 5.491 | 5.823 |
| $I_w$ $[\cdot 10^9 \text{mm}^6]$ | N.A2) | 4.180 | 4.426 | 4.451 | 4.426 |
| $I_y$ $[\cdot 10^6 \text{ mm}^4]$ | N.A2) | 9.110 | 9.373 | 9.373 | 9.373 |
| $I_z$ $[\cdot 10^6 \text{ mm}^4]$ | N.A2) | 1.070 | 1.131 | 1.131 | 1.131 |
| $z_j$ [mm] | N.A2) | 0 1) | 0 | 0 1) | 0 |

1): Not declared by the software but obtained by experimental comparison

2): Not used by the software

3): Not declared by the software

4): $\beta_z$ is given and $z_j$ determined from that

# 5 Presentation and Analysis of Results

## 5.1 Centrically loaded beams

In this section the resulting $M_{cr}$ for the studied cases of centrically loaded beams, obtained using different calculation methods, are compared. The cases of centrically loaded beams are the two studies performed using the monosymmetric I-beam, described in Section 4.1.1, and the study performed using beams with a channel section, described as the former study in Section 4.1.2.

The results are presented using column charts, but numerical results are tabulated in Appendix A.

### 5.1.1 Monosymmetric I-beams

In Figure 5.1 values of $M_{cr}$ for the monosymmetric I-beam described in Section 4.1.1 are presented for various degrees of the end restraint for rotation about the z-axis and warping, $k_z = k_w$, illustrated in Figure 4.2, that are obtained by using different calculation methods.

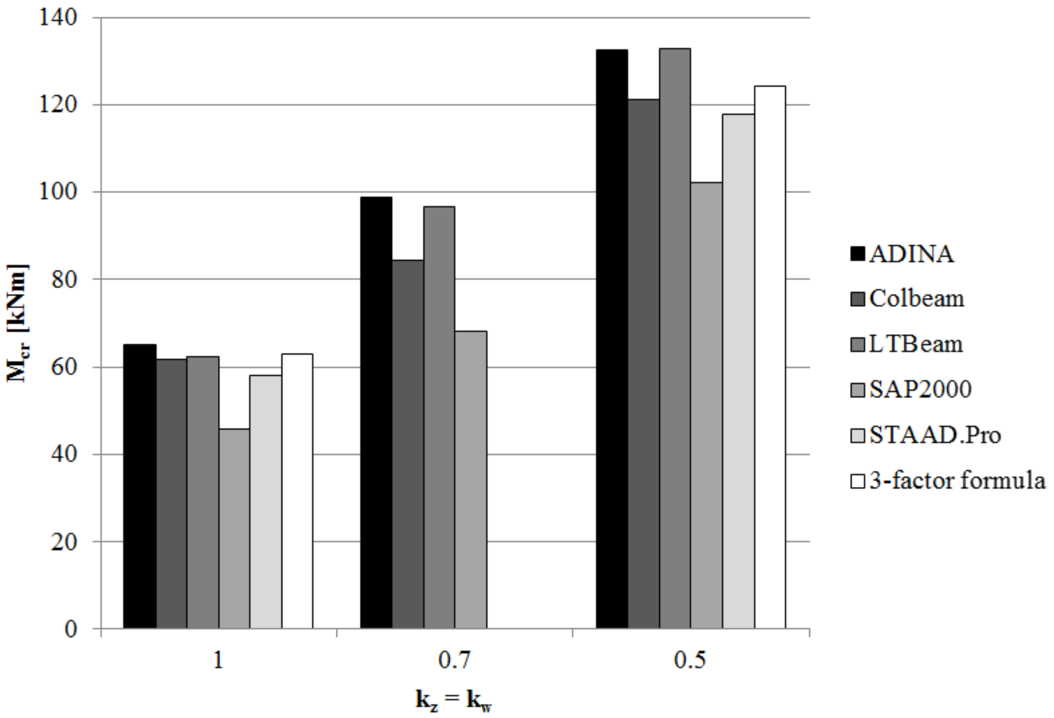



*Figure 5.1 Comparison of the resulting $M_{cr}$ obtained using different methods for the 8 m long monosymmetric I-beam, described in Section 4.1.1, subjected to a concentrated load at mid-span, applied at the centre of gravity, for varied degree of end restraint for rotation about the z-axis and warping, $k_z = k_w$. For clarification, the order of the applied calculation methods, from the top to bottom in the legend, are arranged from left to right in the column chart.*

$M_{cr}$ is not calculated in the case of $k_z = k_w = 0.7$ in STAAD.Pro and hand calculations using the 3-factor formula, since these methods rely on manual input of the C-factors, and tabulated values for the C-factors are only given in the case of either $k_z = k_w = 1$ or $k_z = k_w = 0.5$.

In Figure 5.2 values of $M_{cr}$ for the monosymmetric I-beam described in Section 4.1.1 are presented for various heights of the point of load application, illustrated in Figure 4.3, that are obtained by using different calculation methods.

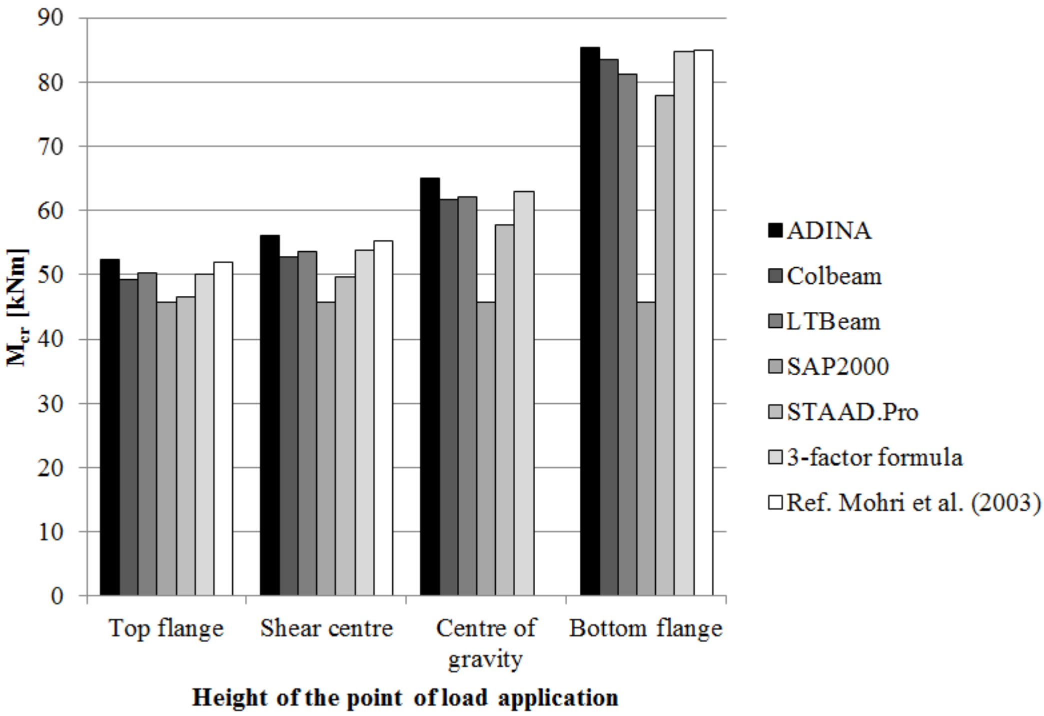



*Figure 5.2* *Comparison of the resulting $M_{cr}$ obtained using different methods for the $8\,m$ long monosymmetric I-beam, described in Section 4.1.1, subjected to a concentrated load at mid-span, with $k_z = k_w = 1$, for varied height of the point of load application. Additional comparison is made with reference values obtained from Mohri et al. (2003). For clarification, the order of the applied calculation methods, from the top to bottom in the legend, are arranged from left to right in the column chart.*

The reference values for $M_{cr}$ in Mohri et al. (2003) were obtained by modelling the monosymmetric I-beam using shell elements in the FEM software Abaqus and solving the eigenvalue problem, similar to what is done in a linearized buckling analysis in ADINA. The reference values are picked from a table in the case of loading at the height of the shear centre and read from graphs in the cases of loading at the height of the top flange and the bottom flange. No reference values are given in Mohri et al. (2003) in the case of loading at the height of the centre of gravity.

The calculations performed by ADINA using the beam element model are based on the sectional parameters presented in Section 4.1.1.

In Colbeam the calculations of $M_{cr}$ are based on built-in sectional properties for the monosymmetric I-section, which is the only difference compared to the case of hand calculations with the 3-factor formula. By comparing the values for the sectional parameters in Colbeam to the ones calculated by hand, shown in Table 4.4, some difference can be seen. The main difference though is in the calculation of the monosymmetry parameter $z_j$, since Colbeam uses the approximate expression in equation (2.34), as mentioned in Section 3.2.1, but the definition in equation (2.33) was used in the hand calculations.

LTBeam uses FEM calculations, like ADINA, to obtain $M_{cr}$, but the difference in the resulting values, even though it is insignificant, might be at least partly caused by different values used for sectional parameters. The chosen modelling procedure allows LTBeam to calculate the sectional properties for the monosymmetric I-section using its own built-in formulas. By comparing the values for the sectional properties in LTBeam to the ones calculated by hand, shown in Table 4.4, some difference can be seen.

The values for $M_{cr}$ obtained from SAP2000 differ from those obtained using the other methods for varied degree of end restraint for rotation about the z-axis and warping, as can be seen in Figure 5.1. As explained in Section 3.4.1, the analytical expression used in SAP2000 to calculate $M_{cr}$ is simpler than the 3-factor formula, and for instance it does not take the degree of monosymmetry of sections explicitly into account. That might be the primary explanation for the difference in $M_{cr}$ compared to results from the other methods in the former study. Additionally the sectional parameters used in SAP2000 are calculated by the software and some values differ as shown in Table 4.4.

$M_{cr}$ obtained from SAP2000 is independent of the vertical position of the PLA, as can be seen in Figure 5.2, as was expected since SAP2000 does not take the vertical position of the loading explicitly into account, as explained in Section 3.4.1.

As discussed in Section 3.5.1, STAAD.Pro does not use the factor $C_3$ when calculating $M_{cr}$ for monosymmetric I-beams, so the monosymmetry parameter $z_j$ in the 3-factor formula is not implemented. Since that ought to be the only difference between calculations using the 3-factor formula by hand and the calculations done by STAAD.Pro, that should explain the difference in the resulting $M_{cr}$ in the two methods.

Calculations by hand using the 3-factor formula, see equation (2.26), are based on the sectional parameters presented in Section 4.1.1 and the C-factors in Table 2.1.

### 5.1.2 Channel beams

In Figure 5.3 values of $M_{cr}$ for beams with the channel section described in Section 4.1.2, subjected to a concentrated mid-span load at the shear centre, are presented for various beam lengths, that are obtained by using different calculation methods.

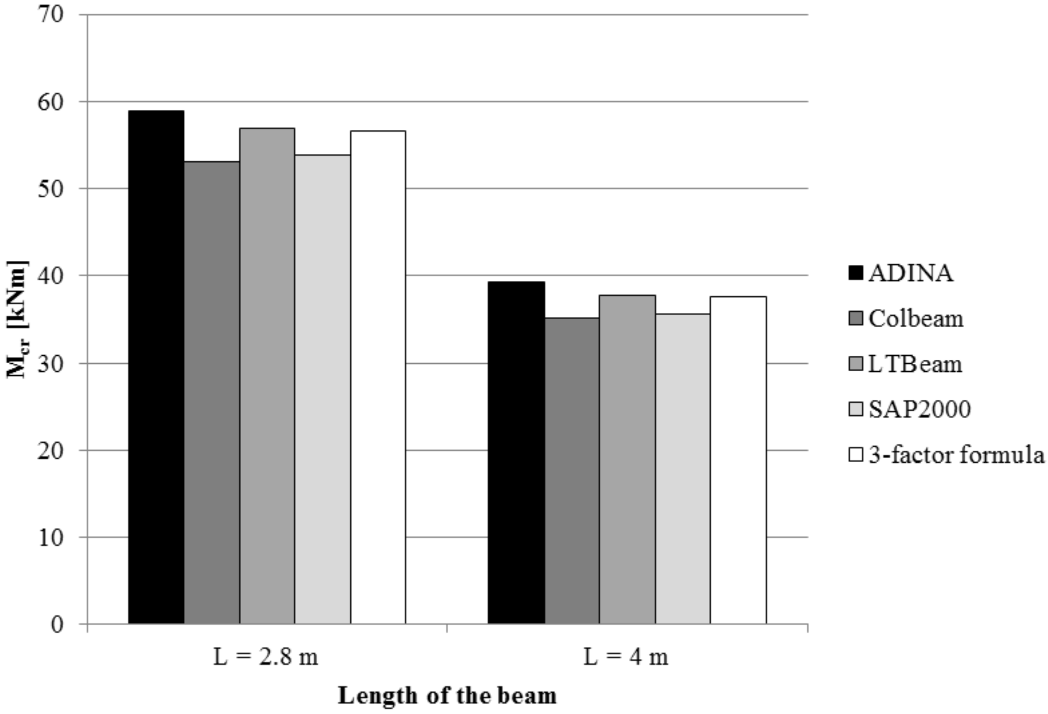



*Figure 5.3* *Comparison of the resulting $M_{cr}$ obtained using different methods for beams with a channel section, described in Section 4.1.2, subjected to a concentrated load at mid-span, applied at the shear centre, for varied beam lengths. For clarification, the order of the applied calculation methods, from the top to bottom in the legend, are arranged from left to right in the column chart.*

In Figure 5.4 values of $M_{cr}$ for beams with the channel section described in Section 4.1.2, subjected to a uniformly distributed load over the whole span at the shear centre, are presented for various beam lengths, that are obtained by using different calculation methods.

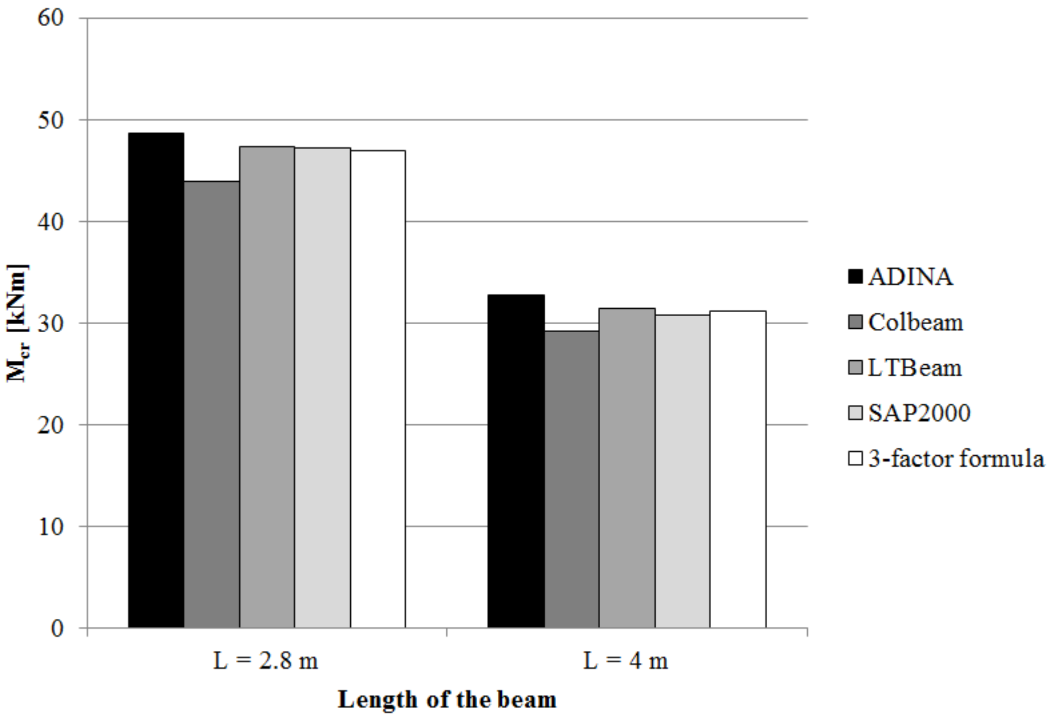



*Figure 5.4* *Comparison of the resulting $M_{cr}$ obtained using different methods for beams with a channel section, described in Section 4.1.2, subjected to a uniformly distributed load over the whole span, applied at the shear centre, for varied beam lengths. For clarification, the order of the applied calculation methods, from the top to bottom in the legend, are arranged from left to right in the column chart.*

Using the FEM calculations with shell elements in ADINA is expected to give the most reasonable results, since it is the only method used in this comparison that takes the channel shape of the section explicitly into account, and the values obtained for $M_{cr}$ might be considered as reference values for comparison with the other methods.

In Colbeam the calculations of $M_{cr}$ are based on the standard UPE160 section, which results in different sectional properties compared to the modified version of the channel section used in the other methods. By comparing the values for the sectional properties in Colbeam to the ones calculated by hand, shown in Table 4.5, the difference can be seen. The different sectional properties are the only difference compared to the case of hand calculations with the 3-factor formula.

The LTBeam calculations of $M_{cr}$ for the channel section are only based on the sectional parameters $I_z$, $I_t$, $I_w$ and $\beta_z$, as described in Section 4.2.2, so the channel shape of the cross section is not taken explicitly into account.

As for the other methods that rely on analytical expressions to calculate $M_{cr}$, the channel shape of the section is not taken explicitly into account by SAP2000. Additionally the sectional parameters used in SAP2000 are calculated by the software and some values differ from those calculated by hand, as shown in Table 4.5.

For informative reasons, calculations by hand using the 3-factor formula, see equation (2.26), were performed even though the formula is not said to be applicable for beams with a section non-symmetric about the minor axis, like the channel section.

As mentioned before, the software STAAD.Pro was not used, since it does not offer a method to evaluate $M_{cr}$ for beams with channel sections (see Section 3.5.1).

## 5.2 Eccentrically loaded channel beams

In this section the resulting relation between bending moment and deformation for the studied cases of elastic, eccentrically loaded beams with a channel section are presented. The results are obtained using an incremental collapse analysis with the LDC method using ADINA, for different magnitude of the eccentricity. The cases of eccentrically loaded beams with a channel section are those presented by the study, described as the latter study in Section 4.1.2.

The results are represented by selected line graphs, but the entire results are presented in Appendix A.

### 5.2.1 Expression of results

When the eccentrically loaded beams are analysed using a collapse analysis, the corresponding centrically loaded beams are also analysed to verify the method and setup of the analysis. Also, the $M_{cr}$ values obtained with a linearized buckling analysis for the centrically loaded cases are presented in the graphs for comparison.

Lateral-torsional buckling of a beam, as the name indicates, results in lateral bending of the beam as well as twisting. By studying the relation between the applied load, or the resulting bending moment, and the deformation of the beam the buckling behaviour can be observed and the critical load evaluated. In the current study it is most appropriate to plot the maximum moment in the beams, i.e. at the centre of the span, as a function of either the lateral displacement or the rotation measured at a chosen point.

For each channel beam the y-displacement and the rotation about the x-axis are measured at different load levels at a node which is located at the centre of the web, at the centre of the span, for all cases. Figure 5.5 illustrates the position of the node.

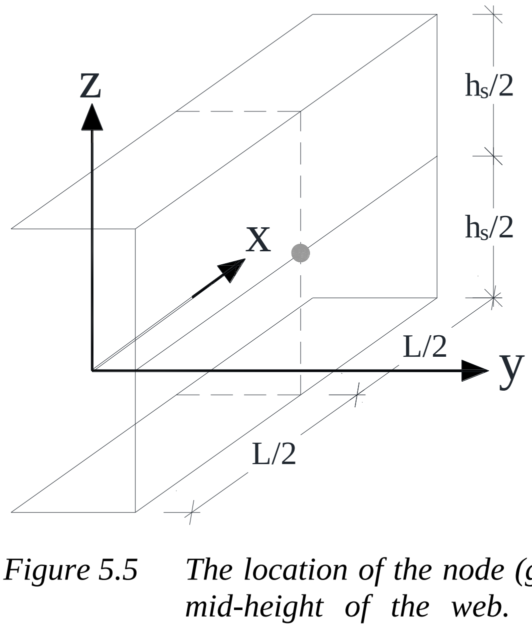



*Figure 5.5 The location of the node (grey) in the mid-span of the beam and at the mid-height of the web. This node is used for measuring the y-displacement and x-rotation at different load levels, for the beams with the channel section, after performing a collapse analysis with the LDC method in ADINA.*

The reason for this choice of a node to be used expressing the results, is that this is the node on the cross section which gives results that are least dependent on each other. The x-rotation is the same in every node on the web in the mid-section and is not dependent on the y-displacement of the beam. On the other hand the y-displacement in the section is dependent on the rotation. As every cross section rotates about its TC (see Section 2.1.6), the translation in the y-direction of the node, at the same height as the TC, is negligible for a very small x-rotation.

### 5.2.2 Relation between bending moment and different kind of displacements

Figure 5.6 and Figure 5.7 present the results from the collapse analysis with the LDC method for a 2.8 m long channel beam subjected to a concentrated load at mid-span. Figure 5.6 shows the maximum moment of the beam plotted against the x-rotation for four different load eccentricities $y_g$, illustrated in Figure 4.5. Also shown is $M_{cr}$ for the corresponding centrically loaded case obtained with a linearized buckling analysis.

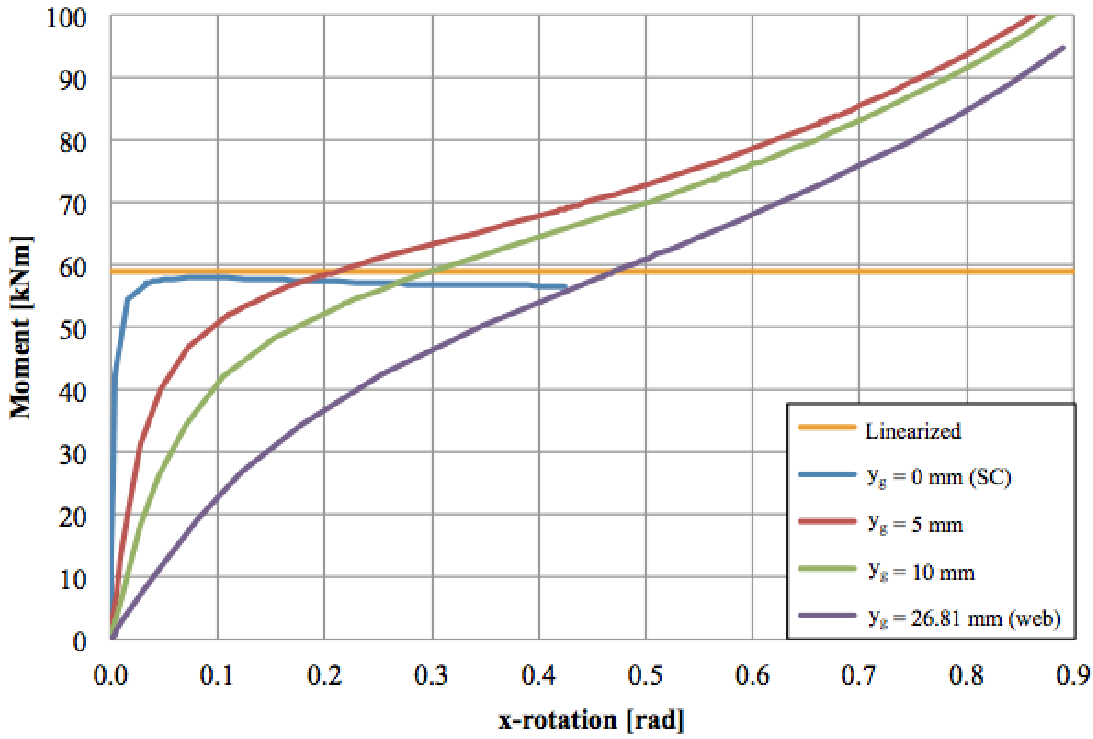



*Figure 5.6 The relationship between the maximum moment, about the major axis, of a 2.8 m long channel beam, subjected to a concentrated load with four different values of the eccentricity, $y_g$, and the rotation about the x-axis for each time step.*

Figure 5.7 shows the maximum moment of the beam plotted against the y-displacement.

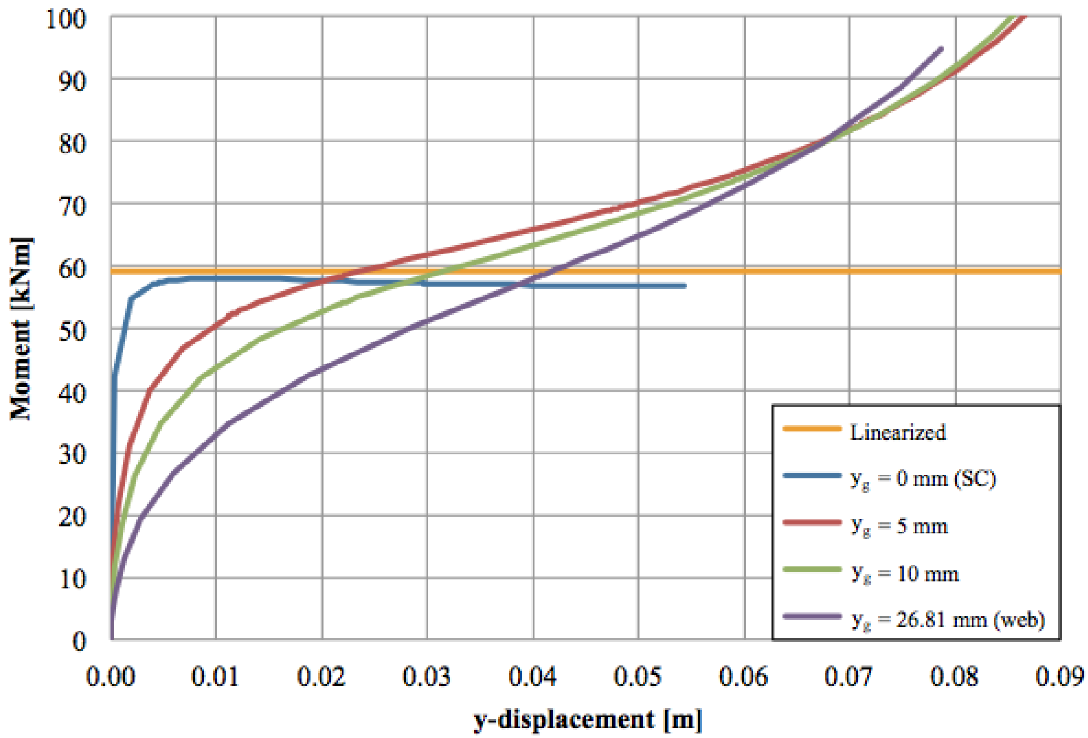



*Figure 5.7* *The relationship between the maximum moment, about the major axis, of a* 2.8 *m long channel beam, subjected to a concentrated load with four different values of the eccentricity,* $y_g$*, and the y-displacement for each time step.*

As Figure 5.6 and Figure 5.7 illustrate, the critical buckling load can clearly be obtained for the centrically loaded beam, which is the case noted with $y_g = 0$ mm. The moment value of the flat part of the curve matches well with the resulting $M_{cr}$ from the linearized buckling analysis, noted as "Linearized" in the graphs. When looking at the curves corresponding to the eccentrically loaded beams, no typical buckling behaviour can be observed. Instead of reaching a certain critical moment value and suddenly buckling laterally like the centrically loaded beam, the eccentrically loaded beams start to deflect in the y-direction early in the loading process. The reason is that the eccentricity creates a concentrated moment about the x-axis at the centre of the span, resulting in a twist as soon as the beam is loaded.

The curves representing the relation between the moment and the y-displacement, shown in Figure 5.7, are round after the y-displacement initiates for the eccentrically loaded beams, and no unstable buckling behaviour can be discerned. The curve representing the relation between the moment and the y-displacement for the centrically loaded beam is flat after the sharp bend at the critical load, but the curves for the eccentrically loaded beams are gradually rising. What happens after that is not of interest in the current thesis and will not be described.

The x-rotation, shown in Figure 5.6, on the other hand, seems to start by linearly increasing with increased loading. The larger the eccentricity is, the more the x-rotation increases. When the load has increased to a certain value, the relationship is no longer linear, but starts to curve. This value is lower for increased eccentricity. Unlike the centrically loaded beam, for which the curve has a sharp bend, the curves of the eccentrically loaded beams are rounder. With increased eccentricity, the more rounded the curve is. When even more load has been applied to the beam, the moment vs. rotation relationship is almost linear again, but with another gradient than before. Further increased loads and x-rotation are not of interest in the current thesis and will not be described.

As the rotation starts immediately after loading, the compression flange translates in the y-direction at the centre from the start as well. As the compression flange is not straight any more, it starts to pull the beam laterally, similar to when the LT-buckling occurs, only now the compression is not large enough to destabilise the beam. As the load increases, more compression and more rotation occur, which results in a significant translation in the y-direction for even a small increase in the load.

### 5.2.3 Influence of the beam length

Figure 5.8 presents a comparison between results for two different lengths of channel beams. This comparison shows the difference between a 2.8 m beam and a 4 m beam subjected to a uniformly distributed load over the whole span at the height of the centre of the web.

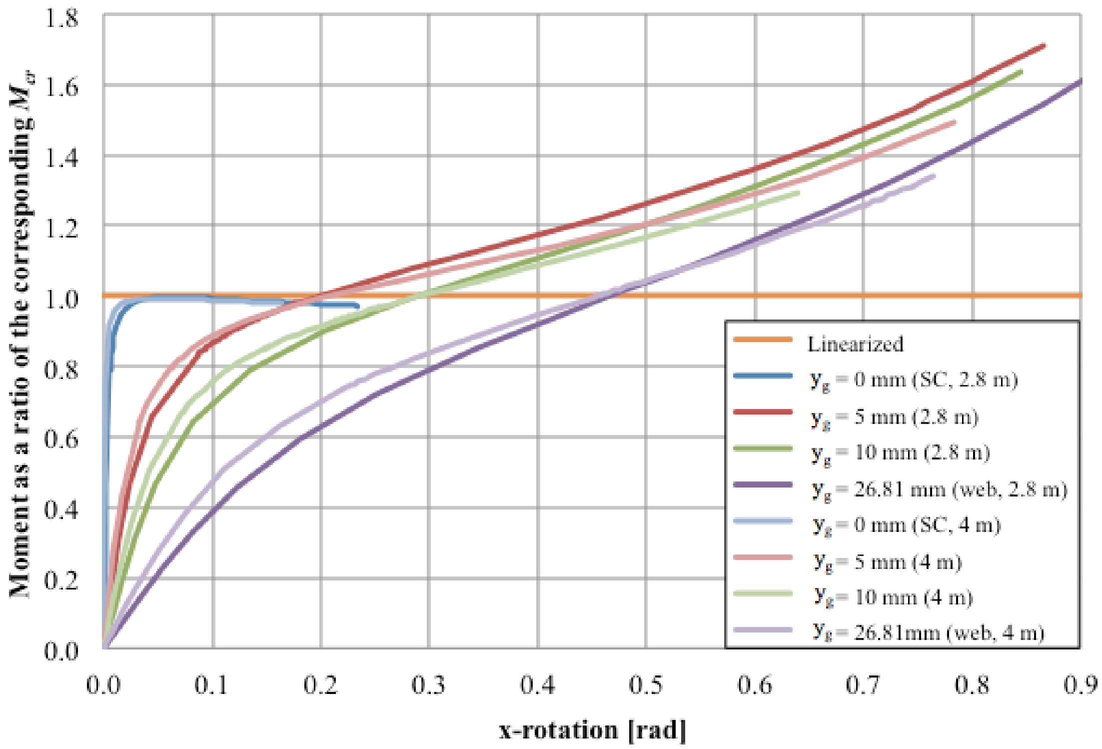



*Figure 5.8* *Comparison between a 2.8 m and a 4 m long channel beams of the relation between their maximum moment about the major axis and the rotation about the x-axis for each time step. The beams are subjected to a uniformly distributed load over the whole span with four different values of the eccentricity, $y_g$. In order to simplify the comparison, the values for the moment for each length are scaled with the $M_{cr}$ value of the corresponding centrically loaded beam, obtained by a linearized buckling analysis.*

By comparing the pairs of curves representing each eccentricity, a difference can be seen between the two beam lengths. When eccentrically loaded, the 4 m beam seems to express the LT-buckling behaviour slightly more than the 2.8 m beam, as the curves are less round and after the bend the gradient of the lines is closer to be flat. This difference could be explained by the bending moments. Considerably less load is needed on the 4 m beam to obtain the same relative moment as in the 2.8 m beam. Less load results in less axial torsion (around the x-axis).

### 5.2.4 Influence of different load eccentricities

Figure 5.9 presents a comparison between various load eccentricities of a 4 m long beam subjected to a uniformly distributed load over the whole span. Also shown is $M_{cr}$ for the corresponding centrically loaded case obtained with a linearized buckling analysis.

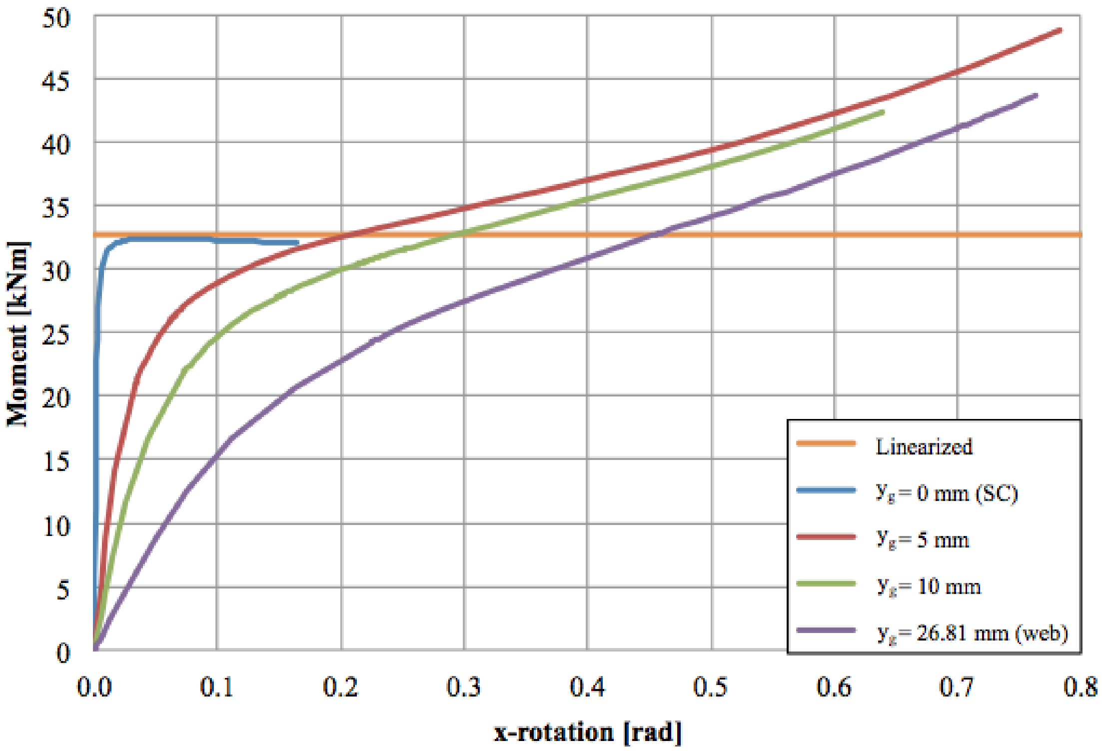



*Figure 5.9 Comparison between the results for four different load positions in the lateral direction, i.e. with different values of the eccentricities, $y_g$, for a 4 m long channel beam, subjected to a uniformly distributed load over the whole span. The curves show the relationship between the maximum moment about the major axis and the rotation about the x-axis for each time step.*

The difference between the results for different eccentricities is obvious. The more the eccentricity is the more each section of the beam rotates, assuming the same load. The sharp bend for the case of the centrically loaded beam is not sharp for the eccentrically loaded ones and is more round as the eccentricity increases. Also, as mentioned before, after the bend in the cases of the eccentrically loaded beams, the curves representing the relations between the moments x-rotation don't become flat, but continue to withstand increased load with increased rotation.

As the eccentricity of the loading increases, more axial torsion acts on the beam, which explains this difference. The beam twists from the beginning, because it is made of elastic material and is subjected to torsion right from the start.

### 5.2.5 Influence of different types of load

Figure 5.10 presents a comparison between two types of load in the form of a relationship between the bending moment and the x-rotation for a 2.8 m long beam.

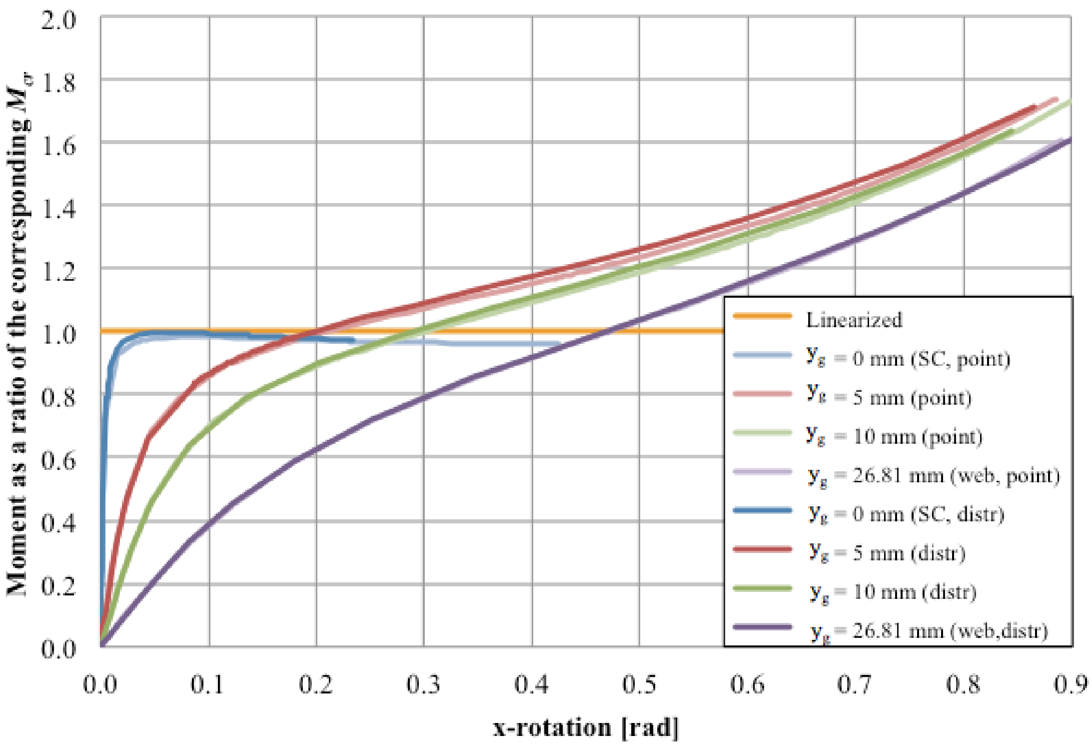



*Figure 5.10 Comparison between the results for a* 2.8 m *long channel beam, when subjected to (1) a concentrated mid-span load and (2) a uniformly distributed load over the whole span. The curves show the relationship between the maximum moment about the major axis and the rotation about the x-axis, for each time step. In both cases different values of the eccentricities, $y_g$, are presented. In order to simplify the comparison, the values for the moment are scaled with the $M_{cr}$ value of the corresponding centrically loaded beam, obtained with a linearized buckling analysis.*

By comparing the pairs of curves representing each eccentricity, no significant difference can be discerned between the results for a 2.8 m beam subjected to a distributed load and the same beam subjected to a point load. All the lines representing the moment-rotation relationship for different eccentricities for a 2.8 m beam subjected to a distributed load, match rather well the ones for the same beam subjected to a point load. The only noticeable difference is when the acting moment is higher than $M_{cr}$ for the corresponding centrically loaded case. Then the beam subjected to a distributed load has to be loaded slightly more to obtain the same x-rotation. The difference is greater when the eccentricity is smaller.

# 6 Final Remarks

## 6.1 Conclusions

In this thesis project the elastic critical moment, $M_{cr}$, has been evaluated for centrically loaded beams with different setups and two types of cross sections; a monosymmetric I-section and a channel section, using five commercial structural engineering software. The behaviour of eccentrically loaded channel beams was studied in order to see if $M_{cr}$ could also be evaluated for such cases.

A monosymmetric I-beam was modelled using five different software tools, subjected to a concentrated load at different load heights with different degree of lateral restraint. Hand calculations using the 3-factor formula were performed for a comparison. The main conclusions drawn from those studies for the monosymmetric I-beam are:

- The obtained results were overall as anticipated, when considering the assumptions made by each software tool and the resulting limitations.
- The degree of lateral restraint affects $M_{cr}$ and is taken into account by all the applied calculation methods.
- The vertical position of the PLA has a significant influence on $M_{cr}$ as expected, with destabilising effects for loading above the SC and vice versa for loading below the SC.
- A good correlation is between the reference values of $M_{cr}$ and the values obtained by the various calculation methods, with the exception of SAP2000, and the obtained results are rather on the safe side.
- SAP2000 does not take the vertical position of the loading into account when calculating $M_{cr}$, only the moment curve, material and sectional properties, and the buckling length.

Beams with a channel section were modelled using four different software tools. Parameters that varied between cases were the lengths of the beams, the eccentricity of the loading and the type of the loading. Hand calculations using the 3-factor formula were performed for a comparison. The main conclusions drawn from those studies are:

- For the cases with a centric load, a good correlation is between the resulting $M_{cr}$ from the various calculation methods, even though some methods were not said to be applicable for this type of a section.
- If the results for the cases with a centric load from the shell element model in ADINA are considered as reference values, the results from all other calculation methods studied give lower values and are thus rather on the safe side.
- With the applied method, no value for $M_{cr}$ could be obtained for the eccentrically loaded beams with a channel section.
- By moving the applied load from a centric location to an eccentric one, twisting/torsion takes over the buckling behaviour of the beams.
- When designing eccentrically loaded channel beams, stability is perhaps not the governing problem, but rather deformations or stresses. The stress problem

is most likely a second order theory stress problem, like a column under eccentric axial load. The eccentricity of the load results in a moment distributed over the column due to the axial load.

## 6.2 Discussion

A few side studies were performed within this thesis project to see if a suitable method could be found to evaluate $M_{cr}$ for an eccentrically loaded channel beam. The attempted methods were not considered applicable or to have a potential of being sufficiently accurate. The methods were not conducted thoroughly, and do thus not play a significant role in this thesis, but will be briefly described here below.

A method based on similar assumptions as the simplified lateral-torsional buckling method in Eurocode 3, described in Section 2.5.1.3 was tried. For an eccentrically loaded channel beam, the compression flange along with $1/3$ of the compressed part of the web was regarded as a column. The bending moment, resulting from the applied load on the beam, was applied as an axial load on the column and the torsional moment, resulting from the eccentricity of the applied load, as a transversal load on the column. Then the column was checked for flexural buckling about the minor axis by performing a linearized buckling analysis in ADINA.

An attempt was made to obtain a clearer buckling behaviour at the first critical load for an eccentrically loaded channel beam. Both the 1$^{st}$ and 2$^{nd}$ buckling mode shapes, resulting from a linearized buckling analysis, were applied to the beam as initial geometric imperfections. The amplitude of the 2$^{nd}$ mode shape imperfection was made around 5-10 times larger than the 1$^{st}$ mode shape imperfection. Then an incremental collapse analysis with the LDC method was performed in ADINA. By doing so, the beam was intended to maintain the 2$^{nd}$ mode shape when loaded, until the critical load of the 1$^{st}$ buckling mode was reached, and then the beam would snap to the 1$^{st}$ mode shape. By examining the resulting load-deformation relation the critical point was expected to be clearer and in particular a large lateral displacement was supposed to occur. This method is considered applicable to employ for centrically loaded beams, but seems not to be relevant for this case.

In an article often referred to in this thesis, Snijder et al. (2008), studied channel beams subjected to an eccentric load. An $M_{cr}$ value for one case is presented, which was determined numerically. In that case a 2.8 m long beam with fork supports and the channel section described in Section 4.1.2 was subjected to a uniformly distributed load at the top of the web. The $M_{cr}$ value was obtained through a linear buckling analysis with the FEM based software Ansys, using shell elements to model the beam and applying the load eccentrically at the intersection of the centre line of the web and the centre line of the top flange (H. H. Snijder, personal correspondence, June 2, 2014).

By using a similar method in ADINA, i.e. performing a linearized buckling analysis of the studied channel beams, modelled as described in Section 4.2.6, with a load applied eccentrically, no stable solution could be obtained. The resulting critical load turned out to be strongly dependent on the magnitude of the applied load, and according to Earls (2007) this type of analysis should be avoided but for the simplest cases of bifurcation buckling.

It was also attempted to apply both the support conditions and the loads to the middle of the web, i.e. modelling a channel beam as described in Section 4.2.6 but moving the supports conditions from points 16 and 18, which represent the shear centre in each end, to points 19 and 21 (see Figure 4.34). The results obtained, and the overall behaviour, from performing an incremental collapse analysis with the LDC method in ADINA, were similar to when the support conditions were applied at the shear centre of the end sections, for eccentric loading.

In Section 2.5 some design methods are described. Section 2.5.2 focuses on eccentrically loaded channel beams. An idea was to find the elastic critical moment for an eccentrically loaded channel beam, which has those design methods implemented. It should be noted that this approximated elastic critical moment for eccentrically loaded channel beams, with the design method proposed by Snijder et al. (2008) implemented, hereafter noted as $M_{cr,c}$, is conservative for eccentricities less than or equal to the distance between the SC and the web of the beam, and is only for the purpose of getting a better understanding of the field of LT-buckling. It could possibly be used in design, as an input in the design methods in Eurocode 3, but cannot be taken as the theoretical elastic critical moment defined in Section 2.4.

$M_{cr}$ is calculated for the eccentrically loaded beam, using FEM since no appropriate analytical expression exists. The corresponding $\bar{\lambda}_{LT}$ is calculated with equation (2.45), and since $\bar{\lambda}_{LT} = \bar{\lambda}_M$ the non-dimensional slenderness parameter $\bar{\lambda}_{MT}$ is calculated with equation (2.62), using the $M_{cr}$ described above. If $\bar{\lambda}_{MT}$ is used as the non-dimensional slenderness in the General case in Eurocode 3 (see Section 2.5.1.1) it results in the following formula:

$$\bar{\lambda}_{MT} = \sqrt{\frac{W_y f_y}{M_{cr,c}}} \tag{6.1}$$

Where $M_{cr,c}$ = Elastic critical moment for eccentrically loaded channel beams taking the design method proposed by Snijder et al. (2008), described in Section 2.5.2, into account

$W_y$ = Appropriate section modulus, depending on the cross section class

$f_y$ = Yield strength of the steel

$\bar{\lambda}_{LT}$ = Non-dimensional slenderness for lateral-torsional buckling

$M_{cr,c}$ can be solved from Equation (6.1) as follows:

$$M_{cr,c} = \frac{W_y f_y}{\bar{\lambda}_{MT}^2} \tag{6.2}$$

Since the elastic critical moment of eccentrically loaded channel beams could not be evaluated in this thesis, this idea was omitted and only briefly discussed here.

## 6.3 Further studies

Here are listed a few examples that the authors think are interesting for further investigation:

- Study further the linearized buckling analysis. In this thesis the analysis in ADINA has proved to give reasonable results for the studied cases involving centric loads, but not when an eccentricity is introduced. Investigate the limits of usage and how accurate it is, and even compare different FEM programs that offers this option.
- Study the stresses and strains that arise when a channel beam is subjected to a torsional moment due to eccentric loading. In this thesis mainly $M_{cr}$ and the overall instability behaviour has been of interest, but this could be a more appropriate approach.
- Study channel beams with more parameters varied than in the present thesis, for example check if the vertical position of the PLA has similar influence on $M_{cr}$ for centrically loaded channel beams as for I-beams.
- Study how real boundary conditions are for channel beams and how they should be applied to different cross section parts of a shell element model.

# 7 References


Access Steel (2005): *NCCI: Mono-symmetrical uniform members under bending and axial compression.* Access Steel. 2006.

Access Steel (2010): *NCCI: Elastic critical moment for lateral torsional buckling.* Access Steel, 2010.

ADINA (2013): *Theory and Modelling Guide – Volume I: ADINA Solids & Structures*. ADINA R&D, Inc., Watertown, MA 02472 USA, December 2013.

ADINA R&D inc. (n.d.): *ADINA – The Company*. (Electronic) Available: <http://www.adina.com/company.shtml> (Accessed: 2014-05-14)

Ahnlén, M., Westlund, J. (2013): *Lateral Torsional Buckling of I-beams. A Parametric Study of Elastic Critical Moments in Structural Design Software*. Chalmers University of Technology, Department of Civil and Environmental Engineering, 2013:59, Göteborg, Sweden, 2013.

Arora J.S. and Wang Q. (n.d.): *53:134 Structural Design II, Design of Beams (Flexural Members) (Part 5 of AISC/LRFD).* (Electronic), Available: <http://user.engineering.uiowa.edu/~design1/StructuralDesignII/BeamDesign.pdf> (Accessed 2014-05-14)

Bentley Systems, Inc. (2011): *STAAD.pro V8i (SELECTseries 3) Help*. Bentley Systems, Inc. Exton, 2011.

Bentley Systems, Inc. (2014a): *3D Structural Analysis & Design Engineering Software – STAAD.Pro*. (Electronic) Available: <http://www.bentley.com/en-US/Products/STAAD.Pro/> (Accessed: 2014-06-04)

Bentley Systems, Inc. (2014b): *http://ftp2.bentley.com/dist/collateral/docs/staad_pro /Staad.Pro_ProductDataSheet_Letter_0314_W.pdf.* (Electronic). Available: <http://ftp2.bentley.com/dist/collateral/docs/staad_pro/Staad.Pro_ProductDataSheet_Letter_0314_W.pdf> (Accessed 2014-06-05)

Canadian Institute of Steel Construction (CISC) (2002): *Torsional Section Properties of Steel Shapes*. Canadian Institute of Steel Construction (CISC), August 2002. Available: <http://www.cisc-icca.ca/files/technical/techdocs/updates/torsionprop.pdf>

CEN (1992): *ENV 1993-1-1:1992 Eurocode 3: Design of steel structures, Part 1: General rules for buildings*, European Committee for Standardization, Brussels, 1992, pp. 268-271.

Chajes A. (1974): *Principles of Structural Stability Theory*, Prentice-Hall Inc. Englewood cliffs, New Jersey, 1974.

Computers & Engineering (2003): *Computers & Engineering:: SAP2000*. (Electronic) Available: <http://www.comp-engineering.com/products/SAP2000/sap2000.html> (Accessed: 2014-05-15)

Computers & Structures Inc. (2011): *Eurocode 3-2005 with Eurocode 8:2004 Steel Frame Design Manual for SAP2000*. Computers and Structures, Inc., Berkeley, California, October, 2011.

CTICM (2012): *LTBeam help*, Centre Technique Industriel de la Construction Métallique Saint-Aubin, France, 2012.

CTICM (n.d.): *LTBeam version 1.0.11 | cticm*. (Electronic) Available: <https://www.cticm.com/content/ltbeam-version-1011> (Accessed: 2014-05-15)

Earls, C.J. (2007): *Observations on eigenvalue buckling analysis within a finite element context*. SSRC Proceedings, New Orleans, Louisiana, USA, 2007. (Electronic), Available: <http://www.slashdocs.com/iunthh/observations-on-eigenvalue-buckling.html> (Accessed 2014-05-14)

EC3 (2005): European Committee for standardization. *EN 1993-1-1:2005. Eurocode 3: Design of Steel Structures. Part 1-1: General Rules and Rules for Buildings*, Brussels, May 2005.

ECCS (2006): *Rules for Member Stability in EN 1993-1-1. Background documentation and design guidelines*. ECCS, Multicomp, Mem Martins, Portugal, 2006, (ISBN: 92-9147-000-84).

Galambos, T.V. (1968): *Structural Members and Frames*. Prentice-Hall Inc., Englewood Cliffs, N.J.

Ziemian R.D. (2010): *Guide to Stability Design Criteria for Metal Structures, Sixth Edition*. John Wiley & Sons, New Jersey, 2010

Hancock G.J. (1996): Design for Distortional Buckling of Flexural Members. *Thin-Walled Structures.* Vol. 27, No. 1 1997. Elsevier Science Ltd., Great Britain, 1996, pp. 3-12.

Hoogenboom, P.C.J., (2006): *Theory of Elasticity, Exams and Answers*. Delft University of Technology, Delft, October, 2006. Available: <http://homepage.tudelft.nl/p3r3s/CT5141_chap7.pdf> (Accessed 2014-06-05)

Höglund, T. (2006): *Design of Steel Structures, Module 6 – Stability of columns and beams*. Swedish Institude of Steel Construction, Luleå University of Technology, Royal Institude of Technology. Skilltryck AB, Publication no., Skillingaryd, Sweden, 2009.

Kitipornchai, S., Wang, C.M. And Trahair, N.S. (1986): Buckling of monosymmetric I-beams under moment gradient. ASCE, *Journal of the Structural Division*, 1986, **112**(ST4), pp 781-799

Kumar, A., Samanta, A. (2005): Distortional buckling in monosymmetric I-beams. *Thin-Walled Structures*. Vol 44, 2006, pp 51-56.

López, A., Yong, D.J., Serna, M. A. (2006): *LATERAL-TORSIONAL BUCKLING OF STEEL BEAMS: A GENERAL EXPRESSION FOR THE MOMENT GRADIENT FACTOR*. Proceeding of the International Colloquium of Stability and Ductility of Steel Structures, D. Camotin et al. (Eds), Lisbon, Portugal, September 6-8, 2006.

Lundh H. (2013): *Hållfasthetslära – Grundläggande hållfasthetslära.* Instant Book AB, Stockholm, 2013.

Mohri, F., Brouki, A., Roth J.C. (2003): Theoretical and numerical stability analyses of unrestrained, mono-symmetric thin-walled beams. *Journal of constructional steel research 59*. Elsevier Science Ltd., pp. 63-90.

NSC (2009): Getting the best out of LTBeam. *New Steel Construction magazine (NSC).* Barrett Byrd Associates, May, 2009. pp. 32-35.

Seah L.K. & Khong P.W. (1990): Lateral-Torsional Buckling of Channel Beams. *J. Construct. Steel Research* 17, 1990. Elsevier Science Publishers Ltd., England, 1991, pp. 265-282.

Serna, M.A., López, A., Puente, I., Yong, D. (2005): Equivalent uniform moment factors for lateral-torsional buckling of steel members. *Journal of Constructional Steel Research*, No. 62, 2006, pp. 566-580.

Snijder, H.H., Hoenderkamp, J. C. D., Bakker M. C. M., Steenbergen, H. M. G. M., de Louw, C. H. M. (2008): Design rules for lateral torsional buckling of channel sections subject to web loading. *Stahlbau*. Vol 77, No. 4, 2008, pp 247-256.

StruProg (2013): *COLBEAM*. (Electronic) Available: <http://www.struprog.se/Colbeam.htm> (Accessed: 2014-05-14)

Timoshenko S.P. and Gere J. (1961): *Theory of Elastic Stability (2nd ed.)*, McGraw-Hill, New York, 1961.

Trahair N.S. (1993): *Flexural-Torsional Buckling of Structures*, CRC Press, Boca Raton, 1993.

# Appendix A Numerical Results

Here are the entire results presented from the studies performed in this thesis.

## A.1 Monosymmetric I-beam

*Table A.1 Comparison of the resulting $M_{cr}$ [kNm] obtained using different methods for the* $8\ m$ *long monosymmetric I-beam, described in Section 4.1.1, subjected to a concentrated load at mid-span, applied at the centre of gravity, for varied degree of end restraint for rotation about the z-axis and warping,* $k_z = k_w$.

| | ADINA | Colbeam | LTBeam | SAP2000 | STAAD.Pro | 3-factor formula |
|---|---|---|---|---|---|---|
| $k_z$=$k_w$=1 | 65.077 | 61.758 | 62.176 | 45.775 | 57.9 | 62.832 |
| $k_z$=$k_w$=0.7 | 98.927 | 84.468 | 96.771 | 68.137 | N.A. | N.A. |
| $k_z$=$k_w$=0.5 | 132.479 | 121.148 | 132.750 | 102.230 | 117.7 | 124.150 |

*Table A.2 Comparison of the resulting $M_{cr}$ [kNm] obtained using different methods for the* $8\ m$ *long monosymmetric I-beam, described in Section 4.1.1, subjected to a concentrated load at mid-span, with* $k_z = k_w = 1$, *for varied height of the point of load application, where TF is the top flange, SC is the shear centre, GC is the centre of gravity and BF is the bottom flange. Additional comparison is made with reference values obtained from Mohri et al. (2003).*

| PLA | ADINA | Colbeam | LTBeam | SAP2000 | STAAD.Pro | 3-factor formula | Mohri et al. (2003) |
|---|---|---|---|---|---|---|---|
| TF | 52.382 | 49.224 | 50.198 | 45.775 | 46.5 | 50.115 | 52.0 |
| SC | 56.046 | 52.774 | 53.662 | 45.775 | 49.7 | 53.723 | 55.32 |
| GC | 65.077 | 61.758 | 62.176 | 45.775 | 57.8 | 62.832 | N.A. |
| BF | 85.255 | 83.390 | 81.238 | 45.775 | 77.8 | 84.673 | 84.9 |

## A.2 Centrically loaded channel beams

*Table A.3 Comparison of the resulting $M_{cr}$ [kNm] obtained using different methods for beams of different lengths with a channel section, described in Section 4.1.2, subjected to a concentrated at mid-span, applied at the shear centre.*

| | ADINA | Colbeam | LTBeam | SAP2000 | 3-factor formula |
|---|---|---|---|---|---|
| $L = 2.8$ m | 58.912 | 53.008 | 56.948 | 53.893 | 56.569 |
| $L = 4$ m | 39.327 | 35.135 | 37.710 | 35.651 | 37.533 |

*Table A.4 Comparison of the resulting $M_{cr}$ [kNm] obtained using different methods for beams of different lengths with a channel section, described in Section 4.1.2, subjected to a uniformly distributed load over the whole span, applied at the shear centre.*

| | ADINA | Colbeam | LTBeam | SAP2000 | 3-factor formula |
|---|---|---|---|---|---|
| $L = 2.8$ m | 48.625 | 43.977 | 47.361 | 47.260 | 46.931 |
| $L = 4$ m | 32.710 | 29.149 | 31.396 | 30.789 | 31.138 |

## A.3 Eccentrically loaded channel beams

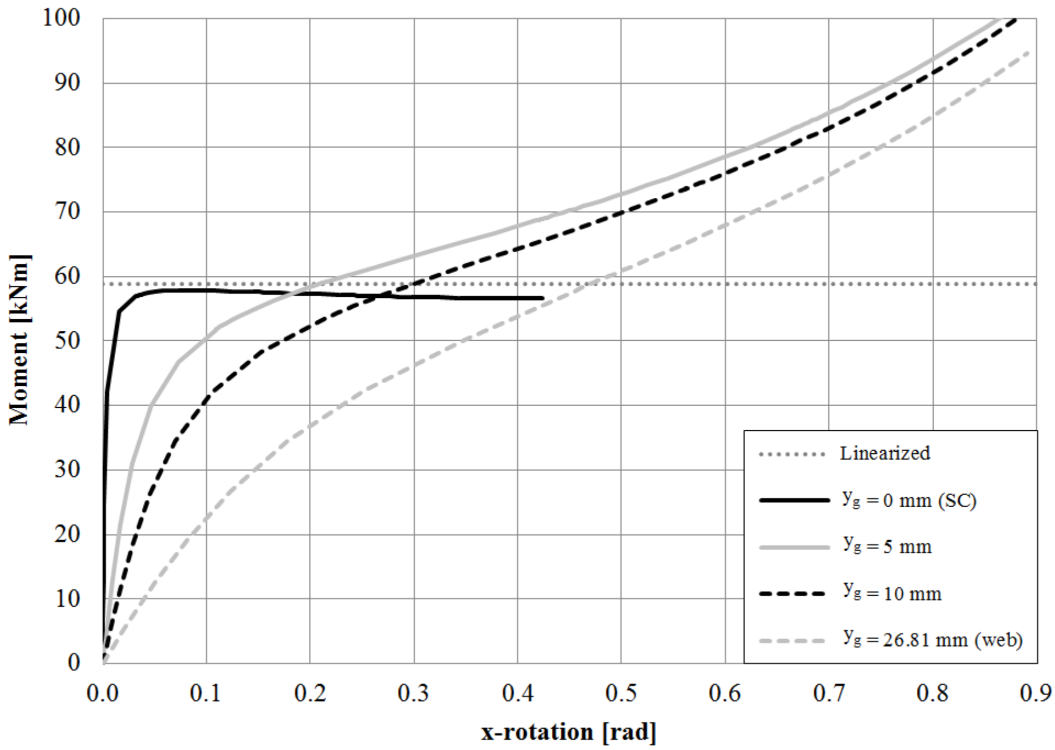
Here are the line graphs presented, representing the relation between the bending moment in eccentrically loaded channel beams and different types of deformations, for different load levels. The results were obtained by performing a collapse analysis with the LDC method in ADINA.

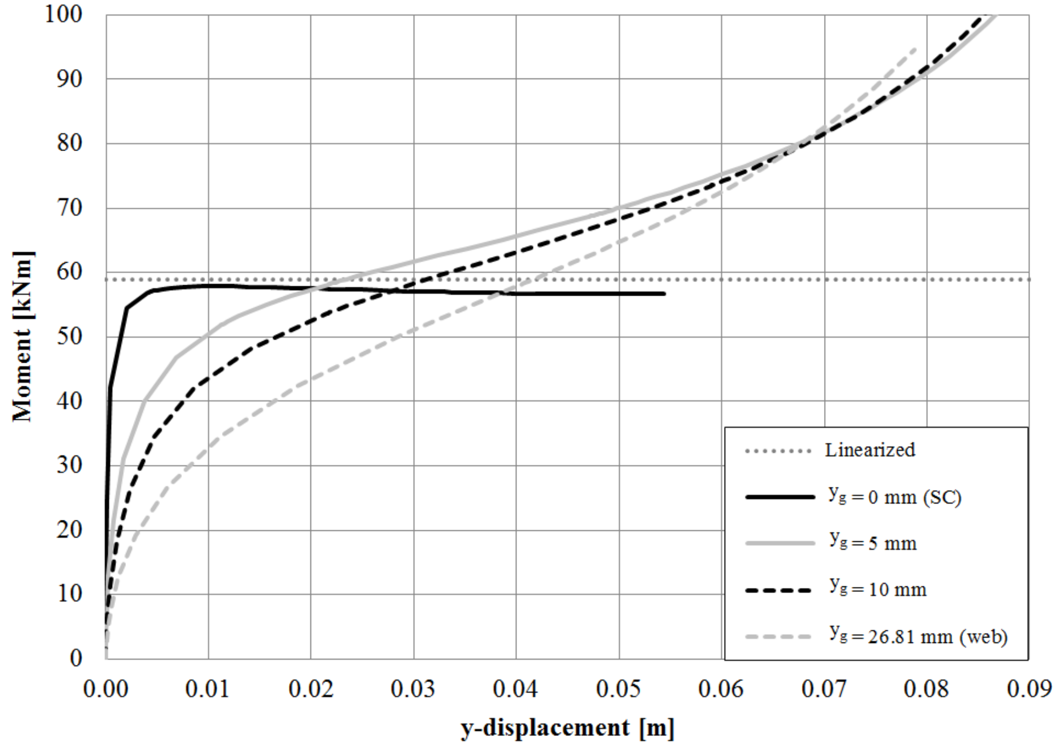
For each case a pair of graphs is presented, one showing the relationship between the bending moment and the x-rotation in the mid-section of the beam and the other the relationship between the bending moment and the y-displacement in the mid-section of the beam.

## A.3.1 Influence of different load eccentricities

### A.3.1.1 2.8 m long beam with a concentrated mid-span load

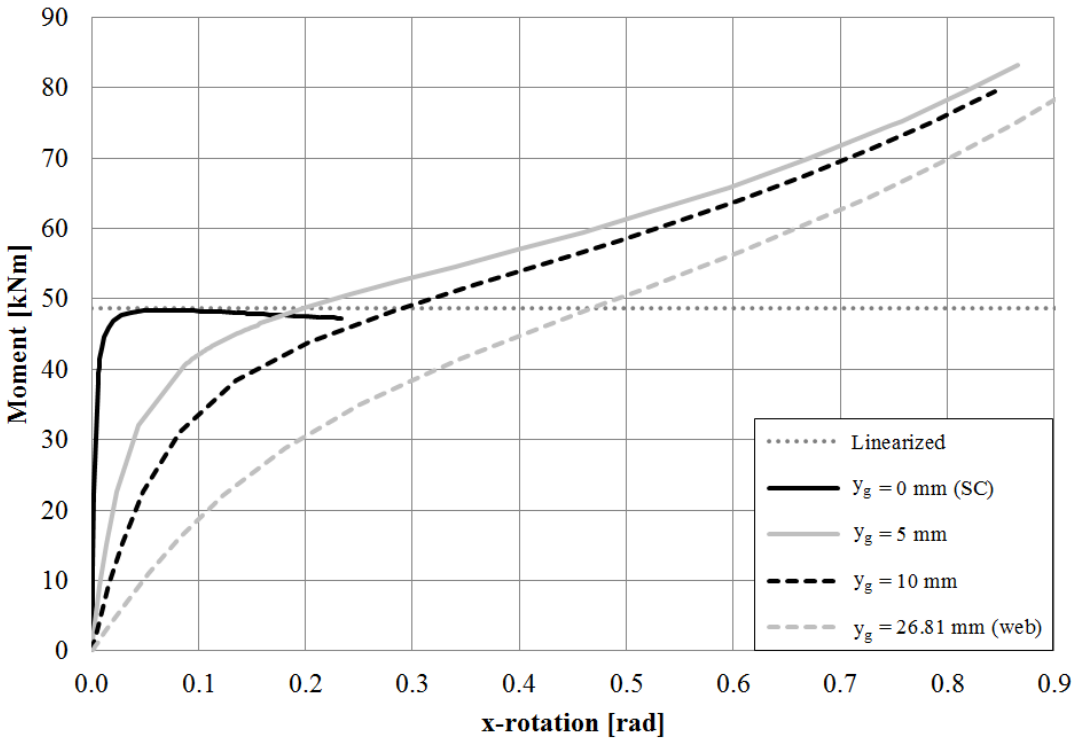
The following figures show the results from the collapse analysis with the LDC method for a 2.8 m long channel beam with the modified UPE160 section subjected to a concentrated mid-span load. Also, the $M_{cr}$ value obtained with a linearized buckling analysis for the centrically loaded case is shown.

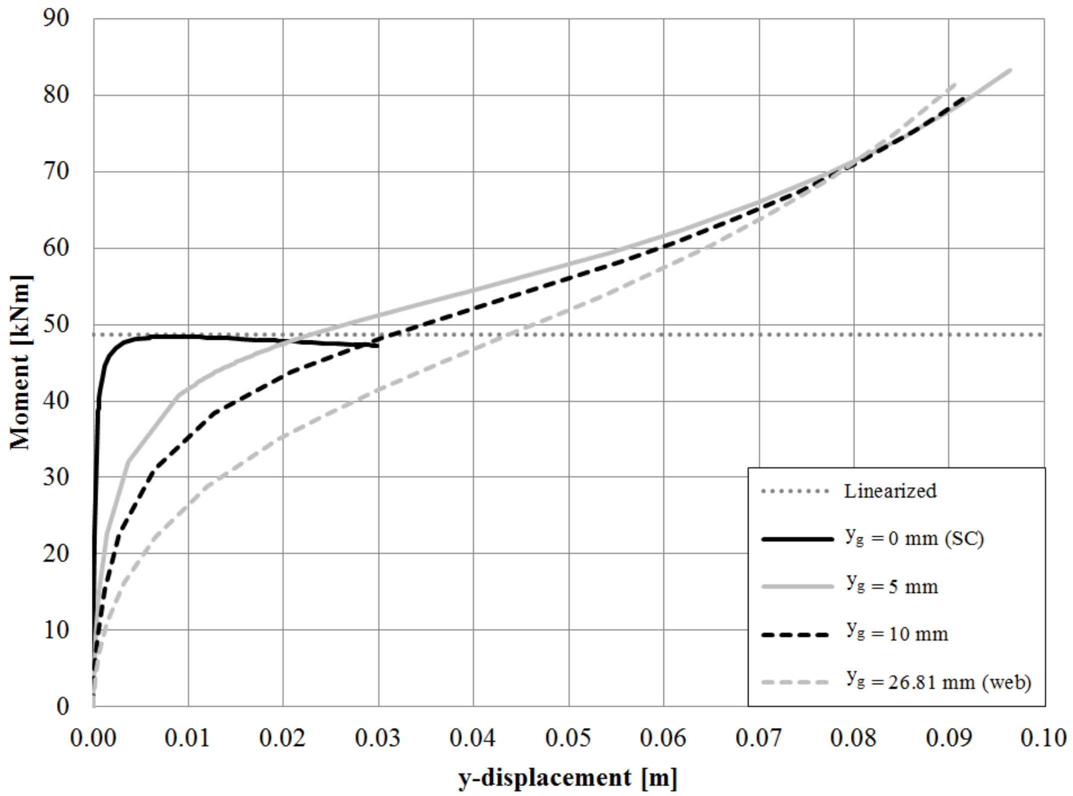


### A.3.1.2 2.8 m long beam with a uniformly distributed load over the whole span

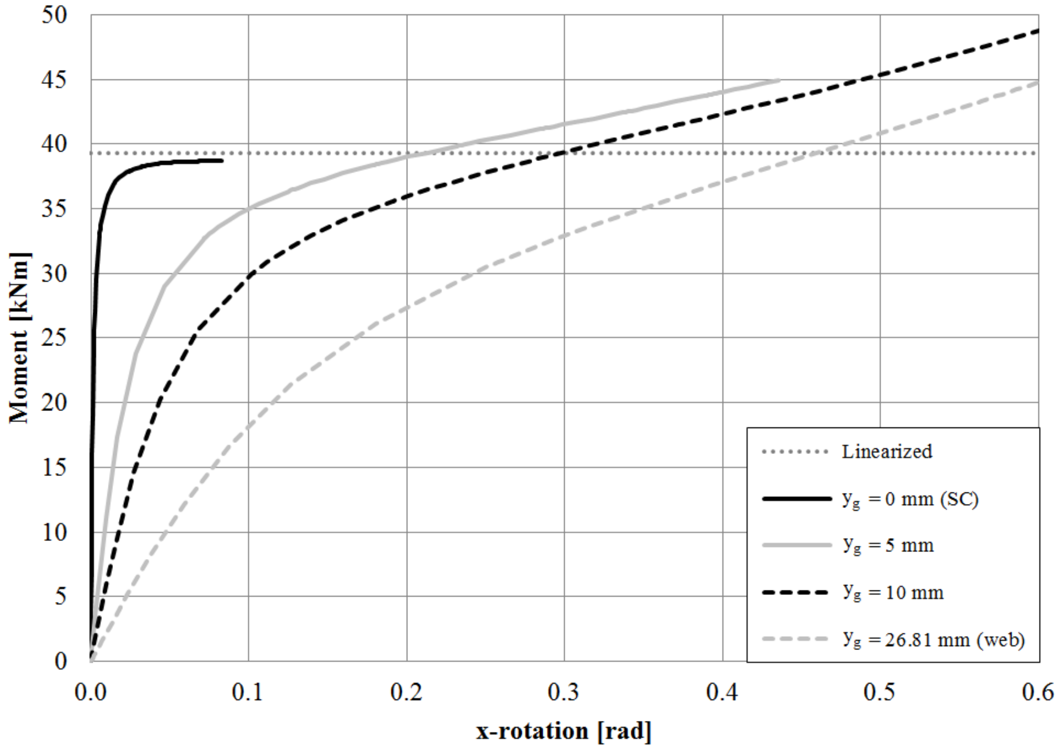
The following figures show the results from the collapse analysis with the LDC method for a 2.8 m long channel beam with the modified UPE160 section subjected to a uniformly distributed load. Also, the $M_{cr}$ value obtained with a linearized buckling analysis for the centrically loaded case is shown.

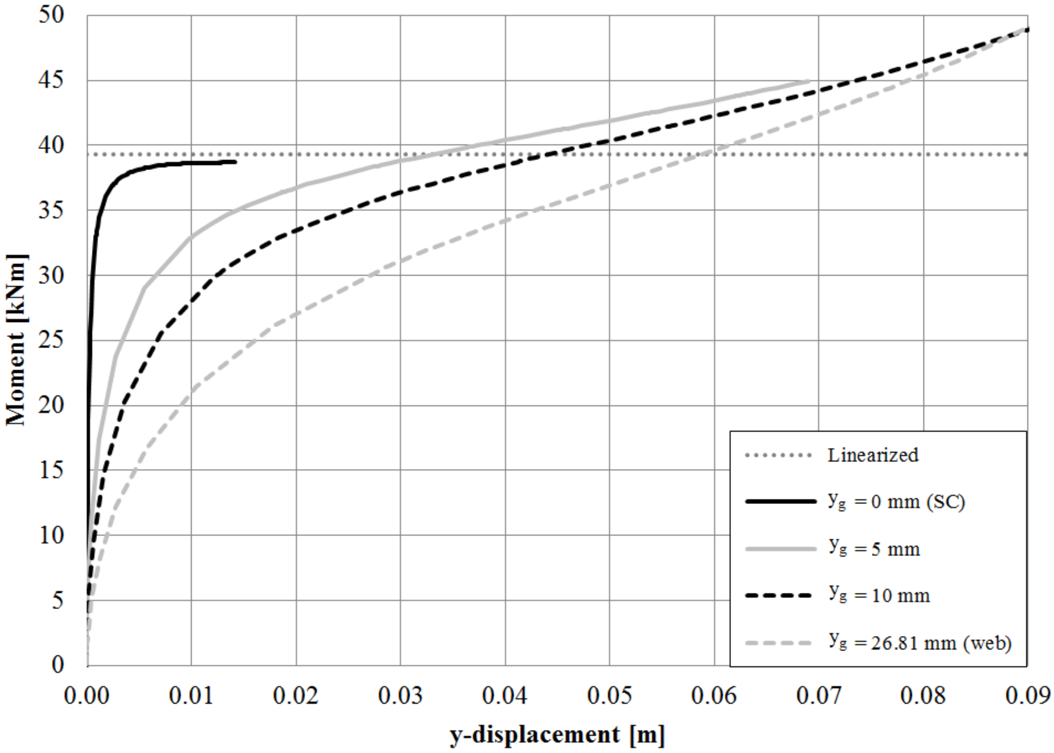


### A.3.1.3 4 m long beam with a concentrated mid-span load

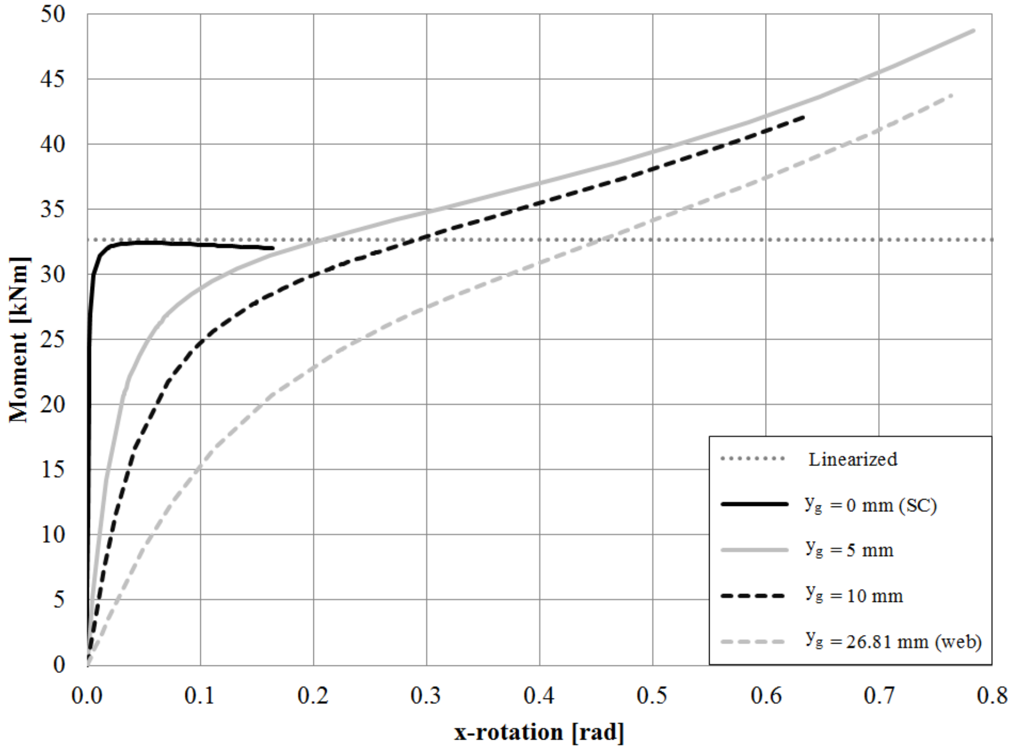
The following figures show the results from the collapse analysis with the LDC method for a 4 m long channel beam with the modified UPE160 section subjected to a concentrated mid-span load. Also, the $M_{cr}$ value obtained with a linearized buckling analysis for the centrically loaded case is shown.

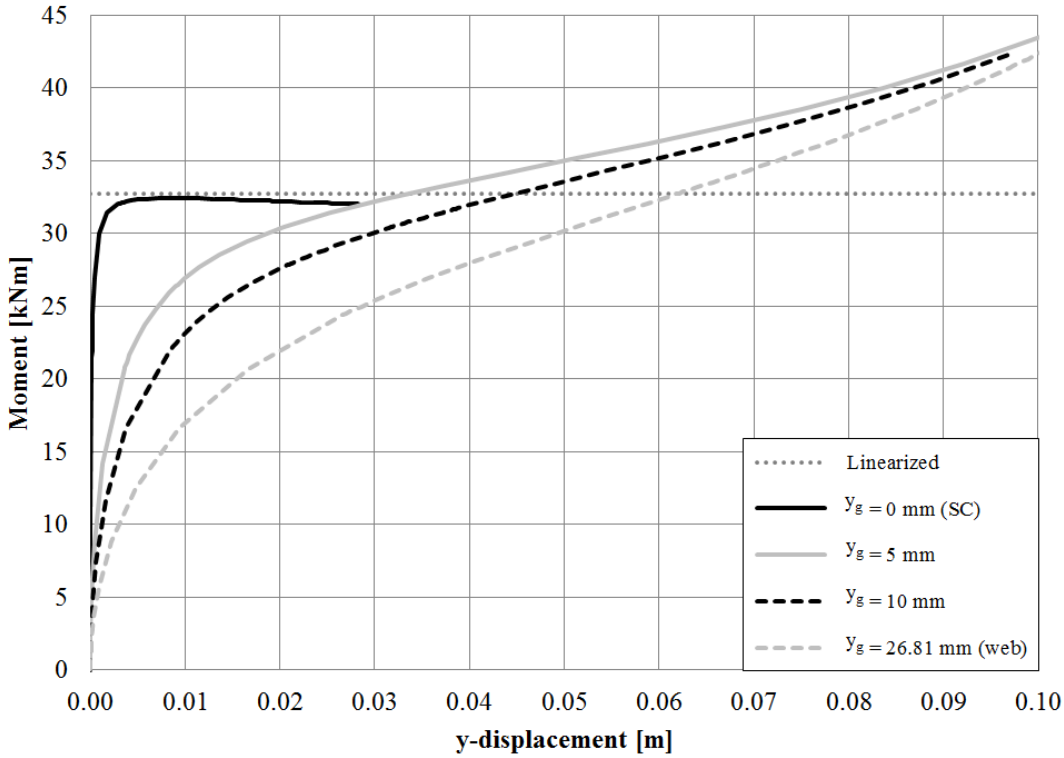


### A.3.1.4 4 m long beam with a uniformly distributed load over the whole span

The following figures show the results from the collapse analysis with the LDC method for a 4 m long channel beam with the modified UPE160 section subjected to a uniformly distributed load. Also, the $M_{cr}$ value obtained with a linearized buckling analysis for the centrically loaded case is shown.

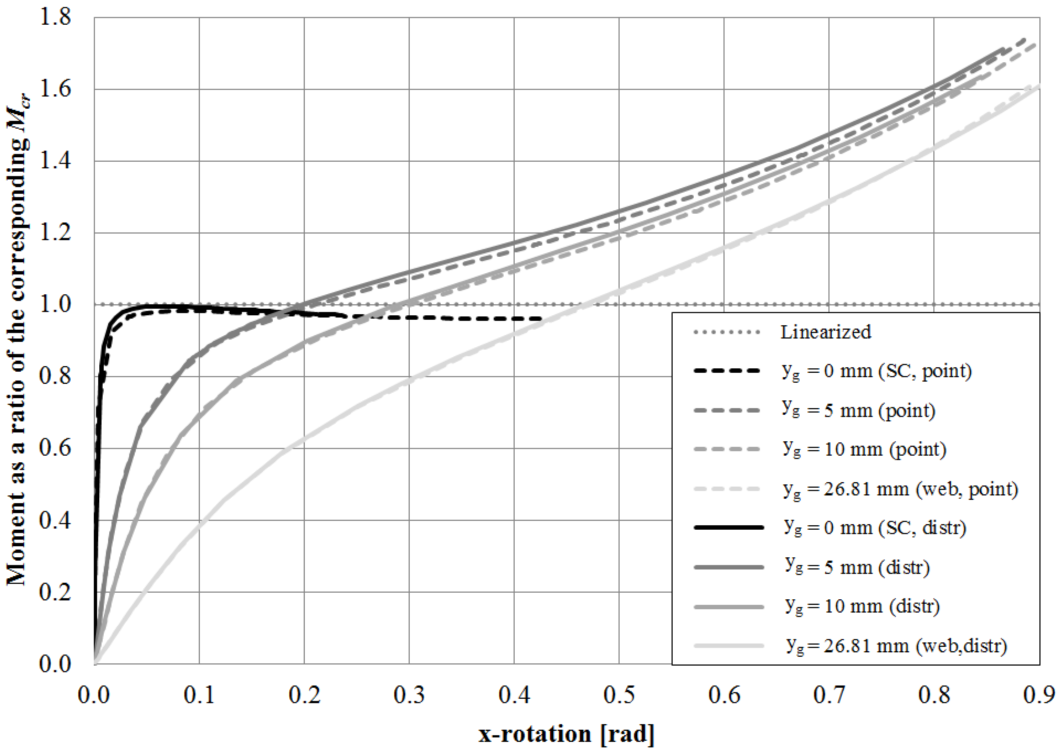


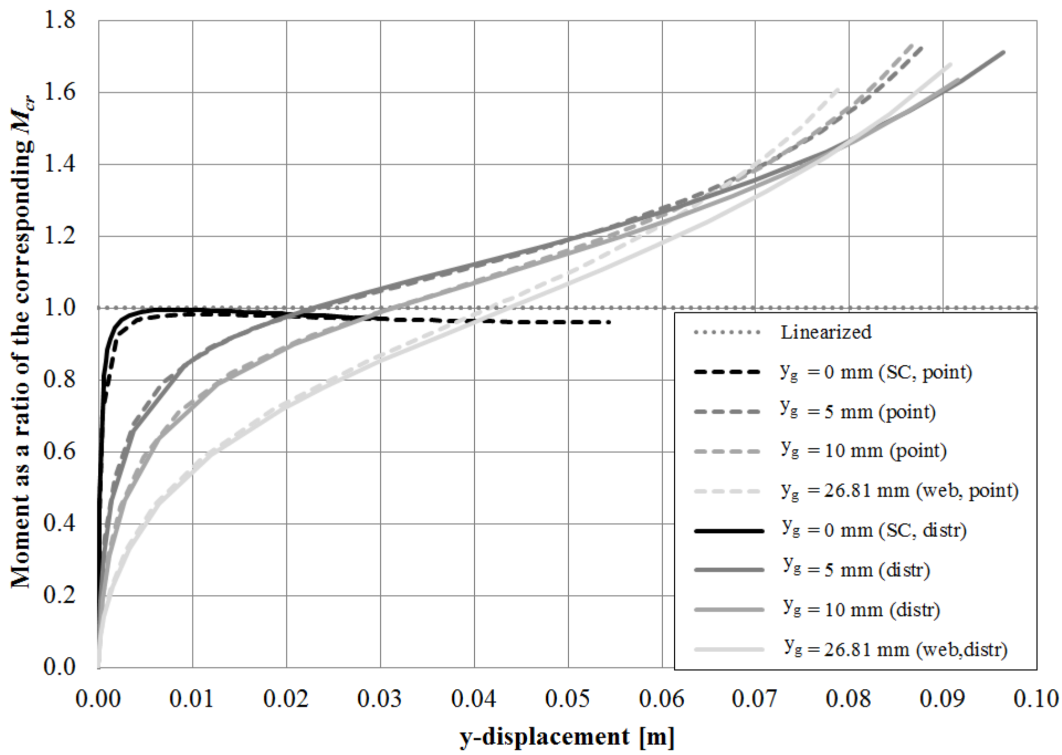



## A.3.2 Influence of different types of load

### A.3.2.1 2.8 m long beam

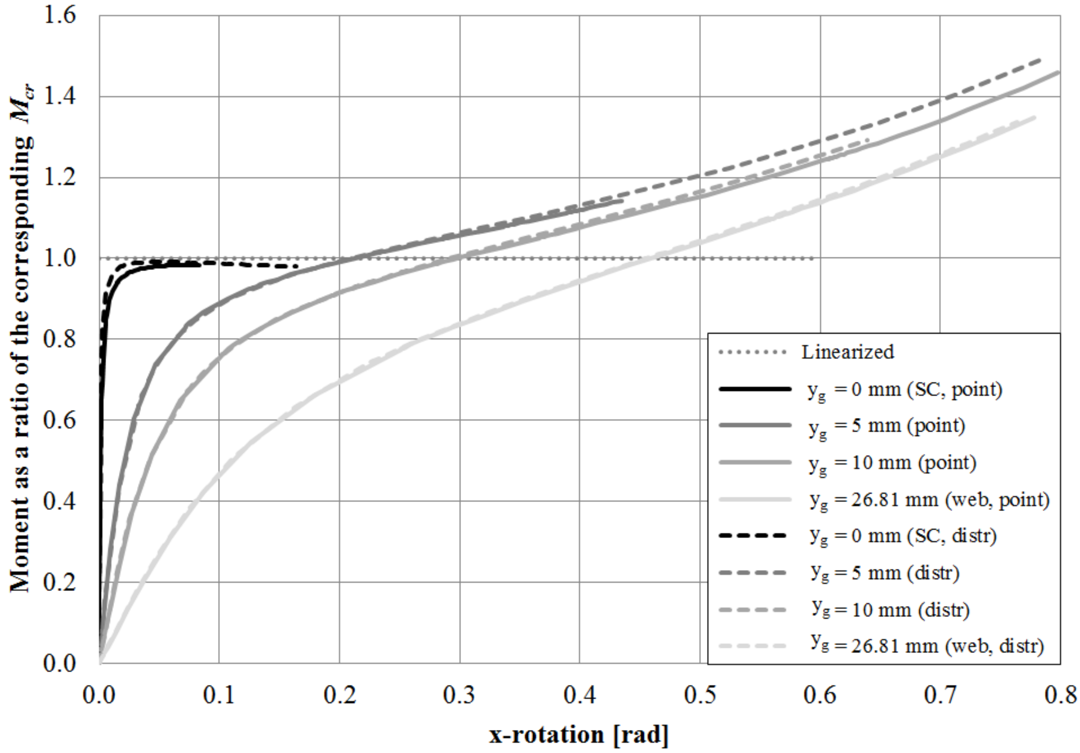
The following figures compare the results from Sections A.3.1.1 and A.3.1.2.

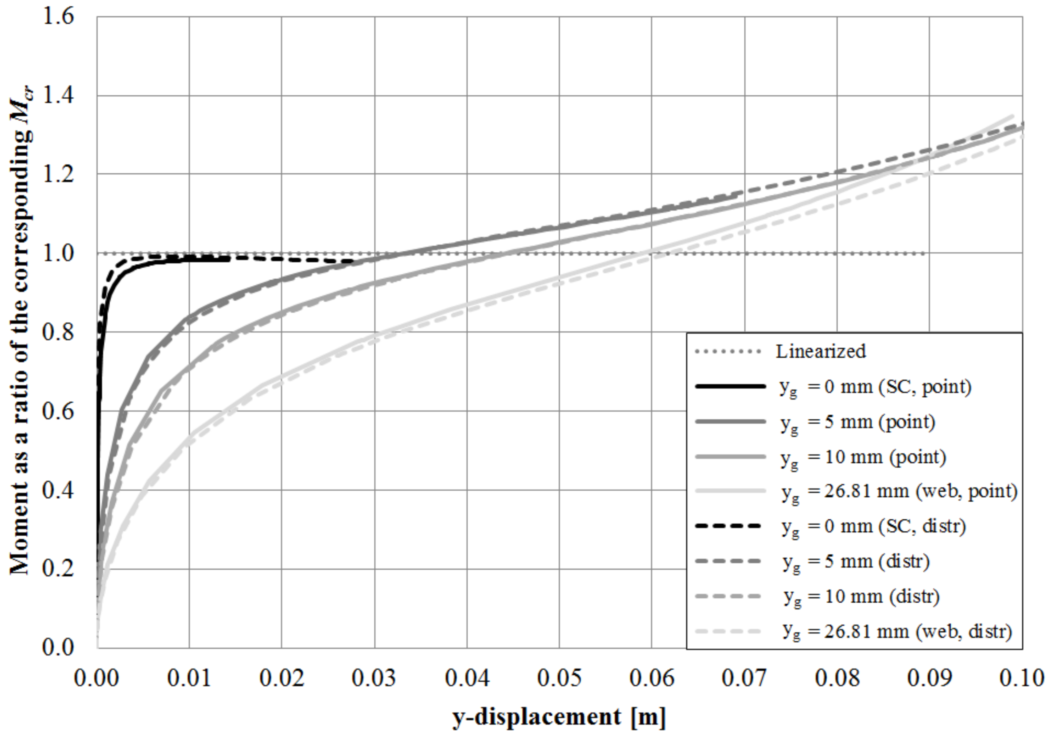


#### A.3.2.2 4 m long beam

The following figures compare the results from Sections A.3.1.3 and A.3.1.4.

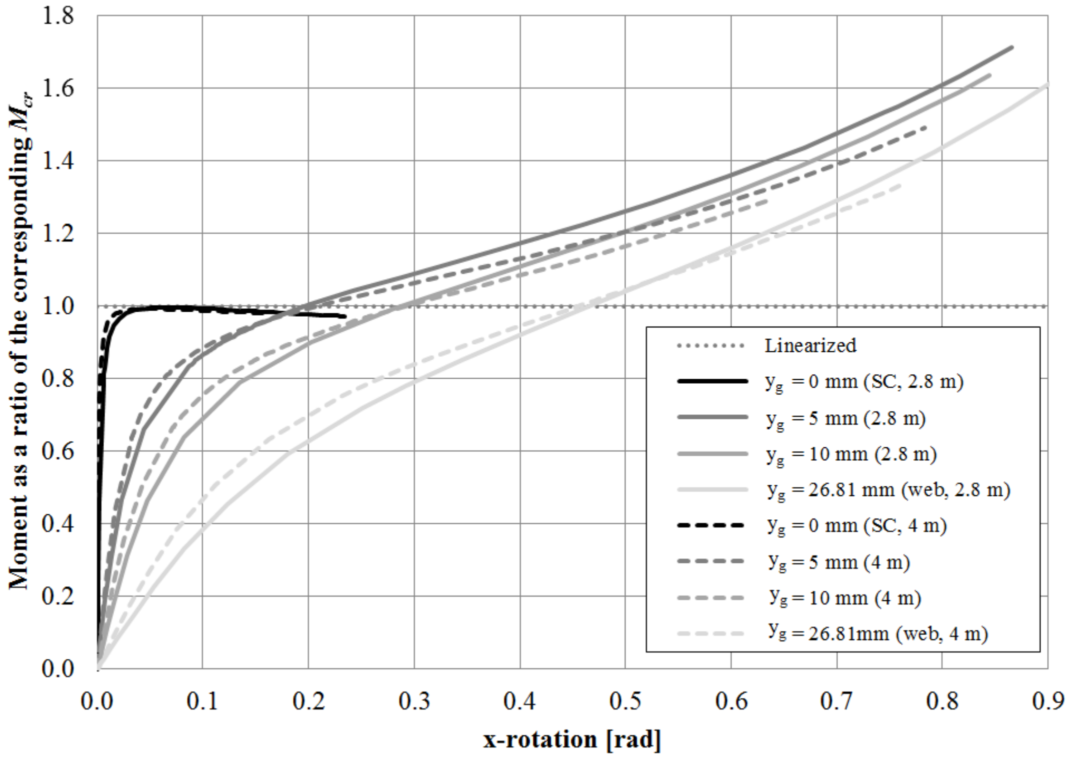


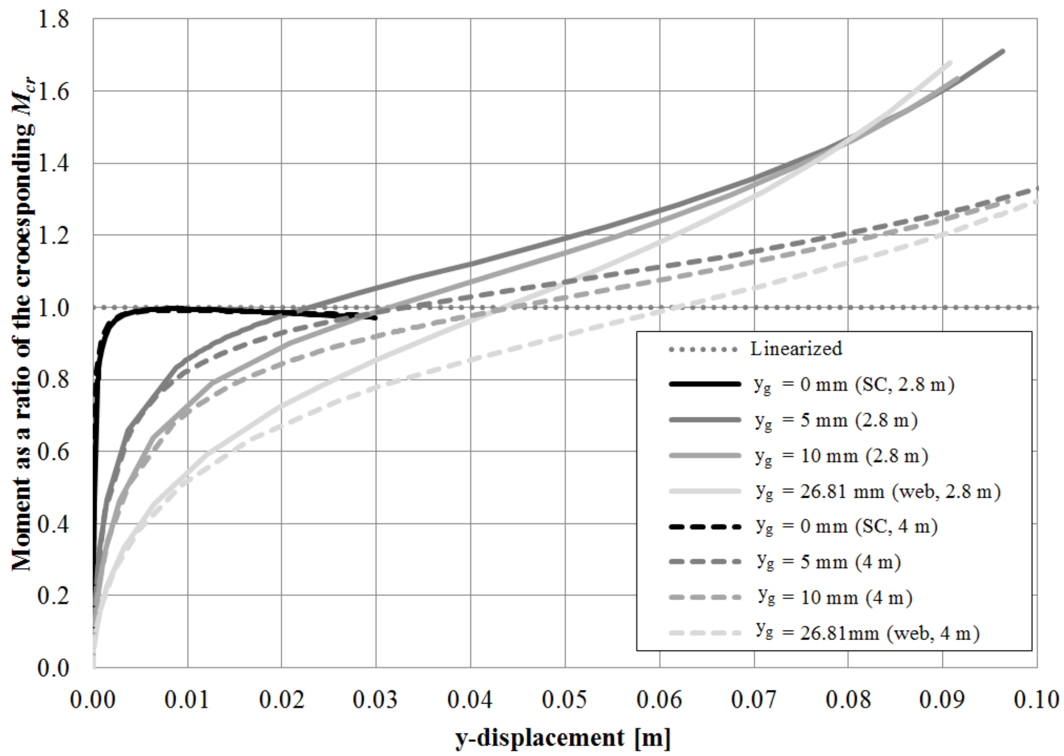



### A.3.3 Influence of the beam length

#### A.3.3.1 Uniformly distributed load over the whole span

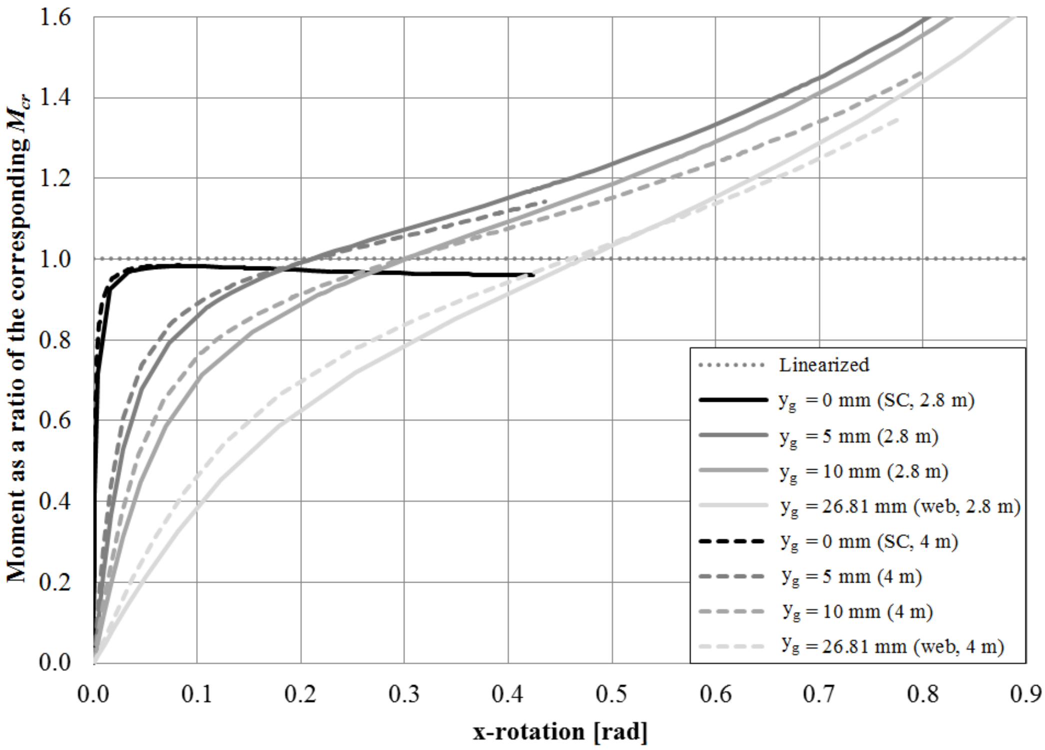
The following figures compare the results from Sections A.3.1.2 and A.3.1.4.

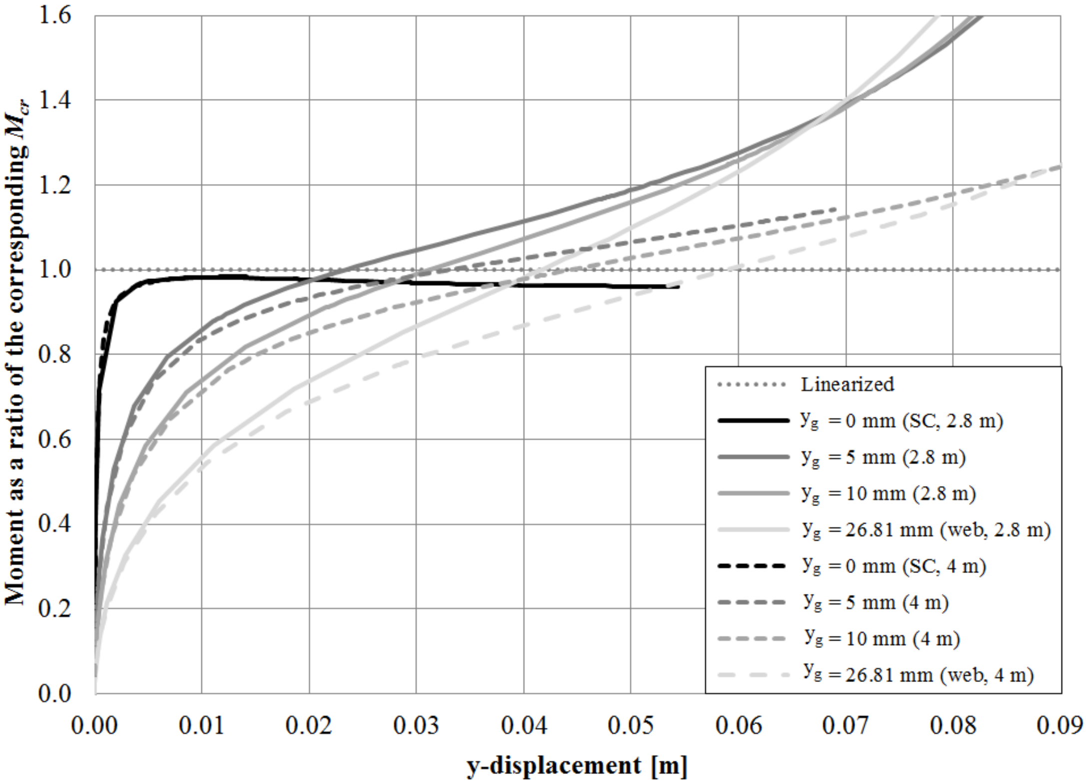


#### A.3.3.2 Concentrated mid-span load

The following figures compare the results from Sections A.3.1.1 and A.3.1.3.



# Appendix B ADINA-In Command Files

Here are selected ADINA-IN Command files presented, that were used to perform the studies in this thesis.

## B.1 Monosymmetric I-beam

Here follows the ADINA-IN Command file used to perform the studies in ADINA, described in Section 4.1.1.

```
************************************************
****                                        ****
****          MONOSYMMETRIC I-BEAM          ****
****      LINEARIZED BUCKLING ANALYSIS      ****
****          GENERAL CROSS SECTION         ****
****           K_Z AND K_W VARIED           ****
****   HEIGHT OF CONCENTRATED LOAD VARIED   ****
****                                        ****
************************************************


************************************************
***************                  ***************
***************    ASSUMPTIONS   ***************
***************                  ***************
************************************************


FEPROGRAM ADINA

KINEMATICS DISPLACE=LARGE STRAINS=SMALL


************************************************
***************                  ***************
***************     GEOMETRY     ***************
***************                  ***************
************************************************


* Comment: Point 4 is the point of load application,
*          with z-coordinate (height):
*          Top flange: 0.123531 (m)
*          Shear centre: 0.086037 (m)
*          Centre of gravity: 0 (m)
*          Bottom flange: -0.176469 (m)

COORDINATES POINT SYSTEM=0
@CLEAR
1 0 0 0 0
2 8 0 0 0
3 4 0 0 0
4 4 0 0.086037 0
@

LINE STRAIGHT NAME=1 P1=1 P2=3
LINE STRAIGHT NAME=2 P1=3 P2=2
```

```
************************************************
***************                  ***************
***************    MATERIALS &   ***************
***************     SECTIONS     ***************
***************                  ***************
************************************************


MATERIAL ELASTIC NAME=1 E=2.1E+11 NU=0.3 MDESCRIP='STEEL'

CROSS-SECTIO PROPERTIES NAME=1 RINERTIA=1.25116E-07,
     SINERTIA=6.01184E-05 TINERTIA=3.39386E-06,
     AREA=0.00438556 CTOFFSET=0.0860365,
     CSOFFSET=0 STINERTI=0,
     SRINERTI=0 TRINERTI=-1.24767E-05,
     WINERTIA=2.8054E-08 WRINERTI=0,
     DRINERTI=5.01858E-06 RRINERTI=9.59754E-05


************************************************
***************                  ***************
***************  ELEMENT GROUPS  ***************
***************                  ***************
************************************************


EGROUP BEAM NAME=1 DISPLACE=LARGE MATERIAL=1,
     SECTION=1 WARP=YES


************************************************
***************                  ***************
***************     ELEMENT      ***************
***************    SUBDIVISION   ***************
***************                  ***************
************************************************


SUBDIVIDE LINE NAME=1 NDIV=50
@CLEAR
2
@

GLINE NODES=2 NCTOLERA=1E-05 GROUP=1 XO=0 YO=1 ZO=0
@CLEAR
1
2
@

GPOINT NODE=102 NCOINCID=ALL NCTOLERA=1E-05
@CLEAR
4
@
```

```
**********************************************
***************                ***************
***************   RIGID LINKS  ***************
***************                ***************
**********************************************


RIGIDLINK NAME=1 SLAVETYP=POINT SLAVENAM=3 MASTERTY=POINT,
     MASTERNA=4 DOFSI=123456


**********************************************
***************                ***************
***************    SUPPORT     ***************
***************   CONDITIONS   ***************
***************                ***************
**********************************************


* Comment: For k_z=k_w=1 don't include
*          'Z-ROTATION'
*          'BEAM-WARP'
*          For k_z=k_w=0.7 include
*          'Z-ROTATION'
*          'BEAM-WARP'
*          in BC1 only.
*          For k_z=k_w=0.5 include
*          'Z-ROTATION'
*          'BEAM-WARP'
*          in both BC1 and BC2.

FIXITY NAME=BC1
@CLEAR
 'X-TRANSLATION'
 'Y-TRANSLATION'
 'Z-TRANSLATION'
 'X-ROTATION'
@

FIXITY NAME=BC2
@CLEAR
 'Y-TRANSLATION'
 'Z-TRANSLATION'
 'X-ROTATION'
@

FIXBOUNDARY POINTS
@CLEAR
1  'BC1'
2  'BC2'
@


**********************************************
***************                ***************
***************     LOADS      ***************
***************                ***************
**********************************************


LOAD FORCE NAME=1 MAGNITUD=28021 FX=0 FY=0 FZ=-1

APPLY-LOAD
@CLEAR
1  'FORCE' 1  'POINT' 4
@
```

```
************************************************
***************                  ***************
***************     ANALYSIS     ***************
***************                  ***************
************************************************


MASTER ANALYSIS=BUCKLING-LOADS MODEX=EXECUTE,
     TSTART=0.00000000000000 IDOF=0 OVALIZAT=NONE,
     FLUIDPOT=AUTOMATIC CYCLICPA=1 IPOSIT=STOP,
     REACTION=YES INITIALS=NO FSINTERA=NO IRINT=DEFAULT,
     CMASS=NO SHELLNDO=AUTOMATIC AUTOMATI=OFF,
     SOLVER=SPARSE CONTACT-=CONSTRAINT-FUNCTION,
     TRELEASE=0.00000000000000 RESTART-=NO FRACTURE=NO,
     LOAD-CAS=NO LOAD-PEN=NO SINGULAR=YES,
     STIFFNES=0.000100000000000000 MAP-OUTP=NONE,
     MAP-FORM=NO NODAL-DE='' POROUS-C=NO ADAPTIVE=0,
     ZOOM-LAB=1 AXIS-CYC=0 PERIODIC=NO VECTOR-S=GEOMETRY,
     EPSI-FIR=NO STABILIZ=NO,
     STABFACT=1.00000000000000E-10 RESULTS=PORTHOLE,
     FEFCORR=NO BOLTSTEP=1 EXTEND-S=YES CONVERT-=NO,
     DEGEN=YES TMC-MODE=NO ENSIGHT-=NO IRSTEPS=1,
     INITIALT=NO TEMP-INT=NO ESINTERA=NO OP2GEOM=NO,
     INSITU-D=NO OP2ERCS=ELEMENT 2DPL-AX=YZ-Z

BUCKLING-LOA NEIGEN=1 METHOD=CLASSICAL
```

## B.2 Centrically loaded channel beams

Here are the ADINA-IN Command files presented used to perform the study, described as the former study in Section 4.1.2, of the 2.8 m long centrically loaded channel beam using a linearized buckling analysis in ADINA.

### B.2.1 Concentrated load

Here follows an ADINA-IN Command file used to perform the study of the 2.8 m long centrically loaded channel beam in ADINA, subjected to a concentrated load at mid-span.

```
************************************************
****                                        ****
****      MODIFIED UPE160 CHANNEL BEAM      ****
****             LENGTH L=2.8M              ****
****      LINEARIZED BUCKLING ANALYSIS      ****
****    CONCENTRATED LOAD IN SHEAR CENTRE   ****
****                                        ****
************************************************


************************************************
***************                  ***************
***************    ASSUMPTIONS   ***************
***************                  ***************
************************************************


FEPROGRAM ADINA

KINEMATICS DISPLACE=LARGE STRAINS=SMALL
```

```
***********************************************
***************                 ***************
***************    GEOMETRY     ***************
***************                 ***************
***********************************************


COORDINATES POINT SYSTEM=0
@CLEAR
1  0    0.01929  0.075 0
2  0    0.01929 -0.075 0
3  1.4  0.01929  0.075 0
4  1.4  0.01929 -0.075 0
5  2.8  0.01929  0.075 0
6  2.8  0.01929 -0.075 0
7  0   -0.04746  0.075 0
8  0   -0.04746 -0.075 0
9  1.4 -0.04746  0.075 0
10 1.4 -0.04746 -0.075 0
11 2.8 -0.04746  0.075 0
12 2.8 -0.04746 -0.075 0
13 0    0        0     0
14 1.4  0        0     0
15 2.8  0        0     0
16 0    0.04610  0     0
17 1.4  0.04610  0     0
18 2.8  0.04610  0     0
19 0    0.01929  0     0
20 1.4  0.01929  0     0
21 2.8  0.01929  0     0
@

SURFACE VERTEX NAME=1 P1=1 P2=19 P3=20 P4=3
SURFACE VERTEX NAME=2 P1=19 P2=2 P3=4 P4=20
SURFACE VERTEX NAME=3 P1=3 P2=20 P3=21 P4=5
SURFACE VERTEX NAME=4 P1=20 P2=4 P3=6 P4=21
SURFACE VERTEX NAME=5 P1=1 P2=7 P3=9 P4=3
SURFACE VERTEX NAME=6 P1=3 P2=9 P3=11 P4=5
SURFACE VERTEX NAME=7 P1=2 P2=8 P3=10 P4=4
SURFACE VERTEX NAME=8 P1=4 P2=10 P3=12 P4=6


***********************************************
***************                 ***************
***************   MATERIALS &   ***************
***************    SECTIONS     ***************
***************                 ***************
***********************************************


MATERIAL ELASTIC NAME=1 E=2.1E+11 NU=0.3 MDESCRIP='STEEL'


***********************************************
***************                 ***************
*************** ELEMENT GROUPS  ***************
***************                 ***************
***********************************************


EGROUP SHELL NAME=1 MATERIAL=1 RINT=2 SINT=2 TINT=7,
     DESCRIPT='WEB' THICKNES=0.0065 TINT-TYP=NEWTON-COTES,
     SHELLTY=STANDARD
```

```
EGROUP SHELL NAME=2 MATERIAL=1 RINT=2 SINT=2 TINT=7,
     DESCRIPT='FLANGES' THICKNES=0.01 TINT-TYP=NEWTON-COTES,
     SHELLTY=STANDARD


*************************************************
***************                   ***************
***************      ELEMENT      ***************
***************    SUBDIVISION    ***************
***************                   ***************
*************************************************



SUBDIVIDE SURFACE NAME=1 NDIV1=3 NDIV2=14
@CLEAR
2
3
4
@

SUBDIVIDE SURFACE NAME=5 NDIV1=4 NDIV2=14
@CLEAR
6
7
8
@

GSURFACE NODES=4 NCTOLERA=1E-05 GROUP=1 PREFSHAP=QUADRILATERAL
@CLEAR
1
2
3
4
@

GSURFACE NODES=4 NCTOLERA=1E-05 GROUP=2 PREFSHAP=QUADRILATERAL
@CLEAR
5
6
7
8
@

GPOINT NODE=436 NCOINCID=ALL NCTOLERA=1E-05
@CLEAR
16
17
18
@



*************************************************
***************                   ***************
***************    RIGID LINKS    ***************
***************                   ***************
*************************************************



* Comment: Two separate rigid links for each end (support condition)
*          to allow warping

RIGIDLINK NAME=1 SLAVETYP=LINE SLAVENAM=1 MASTERTY=POINT,
     MASTERNA=16 DOFSI=123456
@CLEAR
5
@
```

```
RIGIDLINK NAME=2 SLAVETYP=LINE SLAVENAM=13 MASTERTY=POINT,
     MASTERNA=16 DOFSI=23
@CLEAR
18
@

RIGIDLINK NAME=3 SLAVETYP=LINE SLAVENAM=9 MASTERTY=POINT,
     MASTERNA=18 DOFSI=123456
@CLEAR
12
@

RIGIDLINK NAME=4 SLAVETYP=LINE SLAVENAM=17 MASTERTY=POINT,
     MASTERNA=18 DOFSI=23
@CLEAR
22
@

* Comment: Point of load application 'master' and lines in web
*          cross-section 'slave'

RIGIDLINK NAME=5 SLAVETYP=LINE SLAVENAM=7 MASTERTY=POINT,
     MASTERNA=17 DOFSI=123456
@CLEAR
3
@


************************************************
***************                  ***************
***************     SUPPORT      ***************
***************    CONDITIONS    ***************
***************                  ***************
************************************************



FIXITY NAME=BC1
@CLEAR
 'X-TRANSLATION'
 'Y-TRANSLATION'
 'Z-TRANSLATION'
 'X-ROTATION'
@

FIXITY NAME=BC2
@CLEAR
 'Y-TRANSLATION'
 'Z-TRANSLATION'
 'X-ROTATION'
@

FIXBOUNDARY POINTS
@CLEAR
16  'BC1'
18  'BC2'
@
```

```
************************************************
***************                  ***************
***************       LOADS      ***************
***************                  ***************
************************************************


LOAD FORCE NAME=1 MAGNITUD=83615 FX=0 FY=0 FZ=-1

APPLY-LOAD
@CLEAR
1  'FORCE' 1  'POINT' 17
@


************************************************
***************                  ***************
***************     ANALYSIS     ***************
***************                  ***************
************************************************


TOLERANCES ITERATION CONVERGE=EF ETOL=1E-08 RTOL=0.01

MASTER ANALYSIS=BUCKLING-LOADS MODEX=EXECUTE,
     TSTART=0.00000000000000 IDOF=0 OVALIZAT=NONE,
     FLUIDPOT=AUTOMATIC CYCLICPA=1 IPOSIT=STOP,
     REACTION=YES INITIALS=NO FSINTERA=NO IRINT=DEFAULT,
     CMASS=NO SHELLNDO=AUTOMATIC AUTOMATI=OFF,
     SOLVER=SPARSE CONTACT-=CONSTRAINT-FUNCTION,
     TRELEASE=0.00000000000000 RESTART-=NO FRACTURE=NO,
     LOAD-CAS=NO LOAD-PEN=NO SINGULAR=YES,
     STIFFNES=0.000100000000000000 MAP-OUTP=NONE,
     MAP-FORM=NO NODAL-DE='' POROUS-C=NO ADAPTIVE=0,
     ZOOM-LAB=1 AXIS-CYC=0 PERIODIC=NO VECTOR-S=GEOMETRY,
     EPSI-FIR=NO STABILIZ=NO,
     STABFACT=1.00000000000000E-10 RESULTS=PORTHOLE,
     FEFCORR=NO BOLTSTEP=1 EXTEND-S=YES CONVERT-=NO,
     DEGEN=YES TMC-MODE=NO ENSIGHT-=NO IRSTEPS=1,
     INITIALT=NO TEMP-INT=NO ESINTERA=NO OP2GEOM=NO,
     INSITU-D=NO OP2ERCS=ELEMENT 2DPL-AX=YZ-Z

BUCKLING-LOA NEIGEN=1 METHOD=CLASSICAL
```

### B.2.2 Uniformly distributed load

Here follows an ADINA-IN Command file used to perform the study of the 2.8 m long centrically loaded channel beam in ADINA, subjected to a uniformly distributed load over the whole span.

```
************************************************
****                                        ****
****      MODIFIED UPE160 CHANNEL BEAM      ****
****              LENGTH L=2.8M             ****
****      LINEARIZED BUCKLING ANALYSIS      ****
****    DISTRIBUTED LOAD IN SHEAR CENTRE    ****
****                                        ****
************************************************
```

```
************************************************
***************                  ***************
***************    ASSUMPTIONS   ***************
***************                  ***************
************************************************


FEPROGRAM ADINA

KINEMATICS DISPLACE=LARGE STRAINS=SMALL


************************************************
***************                  ***************
***************     GEOMETRY     ***************
***************                  ***************
************************************************


COORDINATES POINT SYSTEM=0
@CLEAR
1  0    0.01929  0.075 0
2  0    0.01929 -0.075 0
3  2.8  0.01929  0.075 0
4  2.8  0.01929 -0.075 0
5  0   -0.04746  0.075 0
6  0   -0.04746 -0.075 0
7  2.8 -0.04746  0.075 0
8  2.8 -0.04746 -0.075 0
9  0    0        0     0
10 2.8  0        0     0
11 0    0.04610  0     0
12 2.8  0.04610  0     0
13 0    0.01929  0     0
14 2.8  0.01929  0     0
@

SURFACE VERTEX NAME=1 P1=1 P2=13 P3=14 P4=3
SURFACE VERTEX NAME=2 P1=13 P2=2 P3=4 P4=14
SURFACE VERTEX NAME=3 P1=1 P2=5 P3=7 P4=3
SURFACE VERTEX NAME=4 P1=2 P2=6 P3=8 P4=4

LINE STRAIGHT NAME=14 P1=11 P2=12


************************************************
***************                  ***************
***************    MATERIALS &   ***************
***************     SECTIONS     ***************
***************                  ***************
************************************************


MATERIAL ELASTIC NAME=1 E=2.1E+11 NU=0.3 MDESCRIP='STEEL'

MATERIAL ELASTIC NAME=2 E=1 NU=0 MDESCRIP='Load line'

CROSS-SECTIO PIPE NAME=1 DIAMETER=0.001 SOLID=YES
```

```
***********************************************
***************                 ***************
***************  ELEMENT GROUPS ***************
***************                 ***************
***********************************************


EGROUP SHELL NAME=1 MATERIAL=1 RINT=2 SINT=2 TINT=7,
     DESCRIPT='WEB' THICKNES=0.0065 TINT-TYP=NEWTON-COTES,
     SHELLTY=STANDARD

EGROUP SHELL NAME=2 MATERIAL=1 RINT=2 SINT=2 TINT=7,
     DESCRIPT='FLANGES' THICKNES=0.01 TINT-TYP=NEWTON-COTES,
     SHELLTY=STANDARD

EGROUP BEAM NAME=3 DISPLACE=LARGE MATERIAL=2,
     DESCRIPT='Load line' SECTION=1


***********************************************
***************                 ***************
***************     ELEMENT     ***************
***************   SUBDIVISION   ***************
***************                 ***************
***********************************************


SUBDIVIDE SURFACE NAME=1 NDIV1=3 NDIV2=28
@CLEAR
2
@

SUBDIVIDE SURFACE NAME=3 NDIV1=4 NDIV2=28
@CLEAR
4
@

SUBDIVIDE LINE NAME=14 NDIV=28

GSURFACE NODES=4 NCTOLERA=1E-05 GROUP=1 PREFSHAP=QUADRILATERAL
@CLEAR
1
2
@

GSURFACE NODES=4 NCTOLERA=1E-05 GROUP=2 PREFSHAP=QUADRILATERAL
@CLEAR
3
4
@

GLINE NODES=2 NCTOLERA=1E-05 GROUP=3 XO=0 YO=1 ZO=0
@CLEAR
14
@


***********************************************
***************                 ***************
***************   RIGID LINKS   ***************
***************                 ***************
***********************************************


* Comment: Two separate rigid links for each end (support condition)
*          to allow warping
```

```
RIGIDLINK NAME=1 SLAVETYP=LINE SLAVENAM=1 MASTERTY=POINT,
     MASTERNA=11 DOFSI=123456
@CLEAR
5
@

RIGIDLINK NAME=2 SLAVETYP=LINE SLAVENAM=8 MASTERTY=POINT,
     MASTERNA=11 DOFSI=23
@CLEAR
11
@

RIGIDLINK NAME=3 SLAVETYP=LINE SLAVENAM=3 MASTERTY=POINT,
     MASTERNA=12 DOFSI=123456
@CLEAR
7
@

RIGIDLINK NAME=4 SLAVETYP=LINE SLAVENAM=10 MASTERTY=POINT,
     MASTERNA=12 DOFSI=23
@CLEAR
13
@

* Comment: Centre line of web 'master' and line of load
*          application 'slave'

RIGIDLINK NAME=5 SLAVETYP=LINE SLAVENAM=14 MASTERTY=LINE,
     MASTERNA=2 DOFSI=123456


************************************************
***************                  ***************
***************     SUPPORT      ***************
***************    CONDITIONS    ***************
***************                  ***************
************************************************



FIXITY NAME=BC1
@CLEAR
 'X-TRANSLATION'
 'Y-TRANSLATION'
 'Z-TRANSLATION'
 'X-ROTATION'
@

FIXITY NAME=BC2
@CLEAR
 'Y-TRANSLATION'
 'Z-TRANSLATION'
 'X-ROTATION'
@

FIXBOUNDARY POINTS
@CLEAR
11  'BC1'
12  'BC2'
@
```

```
***********************************************
***************                ***************
***************      LOADS     ***************
***************                ***************
***********************************************


LOAD LINE NAME=1 MAGNITUD=-49235

APPLY-LOAD
@CLEAR
1  'LINE' 1  'LINE' 14 0 1 0 13 -1 7
@


***********************************************
***************                ***************
***************    ANALYSIS    ***************
***************                ***************
***********************************************


TOLERANCES ITERATION CONVERGE=EF ETOL=1E-08 RTOL=0.01

MASTER ANALYSIS=BUCKLING-LOADS MODEX=EXECUTE,
     TSTART=0.00000000000000 IDOF=0 OVALIZAT=NONE,
     FLUIDPOT=AUTOMATIC CYCLICPA=1 IPOSIT=STOP,
     REACTION=YES INITIALS=NO FSINTERA=NO IRINT=DEFAULT,
     CMASS=NO SHELLNDO=AUTOMATIC AUTOMATI=OFF,
     SOLVER=SPARSE CONTACT-=CONSTRAINT-FUNCTION,
     TRELEASE=0.00000000000000 RESTART-=NO FRACTURE=NO,
     LOAD-CAS=NO LOAD-PEN=NO SINGULAR=YES,
     STIFFNES=0.000100000000000000 MAP-OUTP=NONE,
     MAP-FORM=NO NODAL-DE='' POROUS-C=NO ADAPTIVE=0,
     ZOOM-LAB=1 AXIS-CYC=0 PERIODIC=NO VECTOR-S=GEOMETRY,
     EPSI-FIR=NO STABILIZ=NO,
     STABFACT=1.00000000000000E-10 RESULTS=PORTHOLE,
     FEFCORR=NO BOLTSTEP=1 EXTEND-S=YES CONVERT-=NO,
     DEGEN=YES TMC-MODE=NO ENSIGHT-=NO IRSTEPS=1,
     INITIALT=NO TEMP-INT=NO ESINTERA=NO OP2GEOM=NO,
     INSITU-D=NO OP2ERCS=ELEMENT 2DPL-AX=YZ-Z

BUCKLING-LOA NEIGEN=1 METHOD=CLASSICAL
```

## B.3 Eccentrically loaded channel beams

Here are selected ADINA-IN Command files presented used to perform the study, described as the latter study in Section 4.1.2, of the 2.8 m long channel beam using a collapse analysis with the LDC method in ADINA.

### B.3.1 Concentrated load with an eccentricity of $y_g = 5$ mm

Here follows the two ADINA-IN Command files, one for each type of analysis, used to perform the study of the 2.8 m long channel beam in ADINA, subjected to a concentrated load at mid-span, with an eccentricity of $y_g = 5$ mm. A linearized buckling analysis was performed to obtain the first buckling mode shape, to be used as the initial geometric imperfection in the collapse analysis with the LDC method, as described in Section 4.2.6.

### B.3.1.1 Linearized buckling analysis

```
************************************************
****                                        ****
****      MODIFIED UPE160 CHANNEL BEAM      ****
****            LENGTH L=2.8M               ****
****      LINEARIZED BUCKLING ANALYSIS      ****
****          CONCENTRATED LOAD             ****
****         ECCENTRICITY YG=5MM            ****
****                                        ****
************************************************



************************************************
***************                  ***************
***************   ASSUMPTIONS    ***************
***************                  ***************
************************************************



FEPROGRAM ADINA

KINEMATICS DISPLACE=LARGE STRAINS=SMALL



************************************************
***************                  ***************
***************    GEOMETRY      ***************
***************                  ***************
************************************************



COORDINATES POINT SYSTEM=0
@CLEAR
1  0     0.01929  0.075 0
2  0     0.01929 -0.075 0
3  1.4   0.01929  0.075 0
4  1.4   0.01929 -0.075 0
5  2.8   0.01929  0.075 0
6  2.8   0.01929 -0.075 0
7  0    -0.04746  0.075 0
8  0    -0.04746 -0.075 0
9  1.4  -0.04746  0.075 0
10 1.4  -0.04746 -0.075 0
11 2.8  -0.04746  0.075 0
12 2.8  -0.04746 -0.075 0
13 0     0        0     0
14 1.4   0        0     0
15 2.8   0        0     0
16 0     0.04610  0     0
17 1.4   0.04110  0     0
18 2.8   0.04610  0     0
19 0     0.01929  0     0
20 1.4   0.01929  0     0
21 2.8   0.01929  0     0
@

SURFACE VERTEX NAME=1 P1=1 P2=19 P3=20 P4=3
SURFACE VERTEX NAME=2 P1=19 P2=2 P3=4 P4=20
SURFACE VERTEX NAME=3 P1=3 P2=20 P3=21 P4=5
SURFACE VERTEX NAME=4 P1=20 P2=4 P3=6 P4=21
SURFACE VERTEX NAME=5 P1=1 P2=7 P3=9 P4=3
SURFACE VERTEX NAME=6 P1=3 P2=9 P3=11 P4=5
SURFACE VERTEX NAME=7 P1=2 P2=8 P3=10 P4=4
SURFACE VERTEX NAME=8 P1=4 P2=10 P3=12 P4=6
```

```
**********************************************
***************                ***************
***************   MATERIALS &  ***************
***************    SECTIONS    ***************
***************                ***************
**********************************************


MATERIAL ELASTIC NAME=1 E=2.1E+11 NU=0.3 MDESCRIP='STEEL'


**********************************************
***************                ***************
***************  ELEMENT GROUPS ***************
***************                ***************
**********************************************


EGROUP SHELL NAME=1 MATERIAL=1 RINT=2 SINT=2 TINT=7,
     DESCRIPT='WEB' THICKNES=0.0065 TINT-TYP=NEWTON-COTES,
     SHELLTY=STANDARD


EGROUP SHELL NAME=2 MATERIAL=1 RINT=2 SINT=2 TINT=7,
     DESCRIPT='FLANGES' THICKNES=0.01 TINT-TYP=NEWTON-COTES,
     SHELLTY=STANDARD


**********************************************
***************                ***************
***************     ELEMENT    ***************
***************   SUBDIVISION  ***************
***************                ***************
**********************************************


SUBDIVIDE SURFACE NAME=1 NDIV1=3 NDIV2=14
@CLEAR
2
3
4
@

SUBDIVIDE SURFACE NAME=5 NDIV1=4 NDIV2=14
@CLEAR
6
7
8
@

GSURFACE NODES=4 NCTOLERA=1E-05 GROUP=1 PREFSHAP=QUADRILATERAL
@CLEAR
1
2
3
4
@

GSURFACE NODES=4 NCTOLERA=1E-05 GROUP=2 PREFSHAP=QUADRILATERAL
@CLEAR
5
6
7
8
@
```

```
GPOINT NODE=436 NCOINCID=ALL NCTOLERA=1E-05
@CLEAR
16
17
18
@


************************************************
***************                  ***************
***************    RIGID LINKS   ***************
***************                  ***************
************************************************


* Comment: Two separate rigid links for each end (support condition)
*          to allow warping

RIGIDLINK NAME=1 SLAVETYP=LINE SLAVENAM=1 MASTERTY=POINT,
     MASTERNA=16 DOFSI=123456
@CLEAR
5
@

RIGIDLINK NAME=2 SLAVETYP=LINE SLAVENAM=13 MASTERTY=POINT,
     MASTERNA=16 DOFSI=23
@CLEAR
18
@

RIGIDLINK NAME=3 SLAVETYP=LINE SLAVENAM=9 MASTERTY=POINT,
     MASTERNA=18 DOFSI=123456
@CLEAR
12
@

RIGIDLINK NAME=4 SLAVETYP=LINE SLAVENAM=17 MASTERTY=POINT,
     MASTERNA=18 DOFSI=23
@CLEAR
22
@

* Comment: Point of load application 'master' and lines in web
*          cross-section 'slave'

RIGIDLINK NAME=5 SLAVETYP=LINE SLAVENAM=7 MASTERTY=POINT,
     MASTERNA=17 DOFSI=123456
@CLEAR
3
@


************************************************
***************                  ***************
***************      SUPPORT     ***************
***************    CONDITIONS    ***************
***************                  ***************
************************************************


FIXITY NAME=BC1
@CLEAR
 'X-TRANSLATION'
 'Y-TRANSLATION'
 'Z-TRANSLATION'
 'X-ROTATION'
@
```

```
FIXITY NAME=BC2
@CLEAR
 'Y-TRANSLATION'
 'Z-TRANSLATION'
 'X-ROTATION'
@

FIXBOUNDARY POINTS
@CLEAR
16  'BC1'
18  'BC2'
@


**************************************************
***************                  ***************
***************       LOADS      ***************
***************                  ***************
**************************************************


LOAD FORCE NAME=1 MAGNITUD=40000 FX=0 FY=0 FZ=-1

APPLY-LOAD
@CLEAR
1  'FORCE' 1  'POINT' 17
@


**************************************************
***************                  ***************
***************     ANALYSIS     ***************
***************                  ***************
**************************************************


TOLERANCES ITERATION CONVERGE=EF ETOL=1E-08 RTOL=0.01

MASTER ANALYSIS=BUCKLING-LOADS MODEX=EXECUTE,
     TSTART=0.00000000000000 IDOF=0 OVALIZAT=NONE,
     FLUIDPOT=AUTOMATIC CYCLICPA=1 IPOSIT=STOP,
     REACTION=YES INITIALS=NO FSINTERA=NO IRINT=DEFAULT,
     CMASS=NO SHELLNDO=AUTOMATIC AUTOMATI=OFF,
     SOLVER=SPARSE CONTACT-=CONSTRAINT-FUNCTION,
     TRELEASE=0.00000000000000 RESTART-=NO FRACTURE=NO,
     LOAD-CAS=NO LOAD-PEN=NO SINGULAR=YES,
     STIFFNES=0.000100000000000000 MAP-OUTP=NONE,
     MAP-FORM=NO NODAL-DE='' POROUS-C=NO ADAPTIVE=0,
     ZOOM-LAB=1 AXIS-CYC=0 PERIODIC=NO VECTOR-S=GEOMETRY,
     EPSI-FIR=NO STABILIZ=NO,
     STABFACT=1.00000000000000E-10 RESULTS=PORTHOLE,
     FEFCORR=NO BOLTSTEP=1 EXTEND-S=YES CONVERT-=NO,
     DEGEN=YES TMC-MODE=NO ENSIGHT-=NO IRSTEPS=1,
     INITIALT=NO TEMP-INT=NO ESINTERA=NO OP2GEOM=NO,
     INSITU-D=NO OP2ERCS=ELEMENT 2DPL-AX=YZ-Z

BUCKLING-LOA NEIGEN=1 METHOD=CLASSICAL
```

### B.3.1.2 Collapse analysis with the LDC method

```
************************************************
****                                        ****
****      MODIFIED UPE160 CHANNEL BEAM      ****
****             LENGTH L=2.8M              ****
****  COLLAPSE ANALYSIS WITH THE LDC METHOD ****
****           CONCENTRATED LOAD            ****
****          ECCENTRICITY YG=5MM           ****
****                                        ****
************************************************



************************************************
***************                  ***************
***************    ASSUMPTIONS   ***************
***************                  ***************
************************************************



FEPROGRAM ADINA

KINEMATICS DISPLACE=LARGE STRAINS=SMALL



************************************************
***************                  ***************
***************     GEOMETRY     ***************
***************                  ***************
************************************************



COORDINATES POINT SYSTEM=0
@CLEAR
1  0     0.01929  0.075 0
2  0     0.01929 -0.075 0
3  1.4   0.01929  0.075 0
4  1.4   0.01929 -0.075 0
5  2.8   0.01929  0.075 0
6  2.8   0.01929 -0.075 0
7  0    -0.04746  0.075 0
8  0    -0.04746 -0.075 0
9  1.4  -0.04746  0.075 0
10 1.4  -0.04746 -0.075 0
11 2.8  -0.04746  0.075 0
12 2.8  -0.04746 -0.075 0
13 0     0        0     0
14 1.4   0        0     0
15 2.8   0        0     0
16 0     0.04610  0     0
17 1.4   0.04110  0     0
18 2.8   0.04610  0     0
19 0     0.01929  0     0
20 1.4   0.01929  0     0
21 2.8   0.01929  0     0
@

SURFACE VERTEX NAME=1 P1=1 P2=19 P3=20 P4=3
SURFACE VERTEX NAME=2 P1=19 P2=2 P3=4 P4=20
SURFACE VERTEX NAME=3 P1=3 P2=20 P3=21 P4=5
SURFACE VERTEX NAME=4 P1=20 P2=4 P3=6 P4=21
SURFACE VERTEX NAME=5 P1=1 P2=7 P3=9 P4=3
SURFACE VERTEX NAME=6 P1=3 P2=9 P3=11 P4=5
SURFACE VERTEX NAME=7 P1=2 P2=8 P3=10 P4=4
SURFACE VERTEX NAME=8 P1=4 P2=10 P3=12 P4=6
```

```
************************************************
***************                 ***************
***************   MATERIALS &   ***************
***************    SECTIONS     ***************
***************                 ***************
************************************************


MATERIAL ELASTIC NAME=1 E=2.1E+11 NU=0.3 MDESCRIP='STEEL'


************************************************
***************                 ***************
*************** ELEMENT GROUPS  ***************
***************                 ***************
************************************************


EGROUP SHELL NAME=1 MATERIAL=1 RINT=2 SINT=2 TINT=7,
     DESCRIPT='WEB' THICKNES=0.0065 TINT-TYP=NEWTON-COTES,
     SHELLTY=STANDARD

EGROUP SHELL NAME=2 MATERIAL=1 RINT=2 SINT=2 TINT=7,
     DESCRIPT='FLANGES' THICKNES=0.01 TINT-TYP=NEWTON-COTES,
     SHELLTY=STANDARD


************************************************
***************                 ***************
***************     ELEMENT     ***************
***************   SUBDIVISION   ***************
***************                 ***************
************************************************


SUBDIVIDE SURFACE NAME=1 NDIV1=3 NDIV2=14
@CLEAR
2
3
4
@

SUBDIVIDE SURFACE NAME=5 NDIV1=4 NDIV2=14
@CLEAR
6
7
8
@

GSURFACE NODES=4 NCTOLERA=1E-05 GROUP=1 PREFSHAP=QUADRILATERAL
@CLEAR
1
2
3
4
@

GSURFACE NODES=4 NCTOLERA=1E-05 GROUP=2 PREFSHAP=QUADRILATERAL
@CLEAR
5
6
7
8
@
```

```
GPOINT NODE=436 NCOINCID=ALL NCTOLERA=1E-05
@CLEAR
16
17
18
@

***********************************************
***************                 ***************
***************   RIGID LINKS   ***************
***************                 ***************
***********************************************


* Comment: Two separate rigid links for each end (support condition)
*          to allow warping

RIGIDLINK NAME=1 SLAVETYP=LINE SLAVENAM=1 MASTERTY=POINT,
     MASTERNA=16 DOFSI=123456
@CLEAR
5
@

RIGIDLINK NAME=2 SLAVETYP=LINE SLAVENAM=13 MASTERTY=POINT,
     MASTERNA=16 DOFSI=23
@CLEAR
18
@

RIGIDLINK NAME=3 SLAVETYP=LINE SLAVENAM=9 MASTERTY=POINT,
     MASTERNA=18 DOFSI=123456
@CLEAR
12
@

RIGIDLINK NAME=4 SLAVETYP=LINE SLAVENAM=17 MASTERTY=POINT,
     MASTERNA=18 DOFSI=23
@CLEAR
22
@

* Comment: Point of load application 'master' and lines in web
*          cross-section 'slave'

RIGIDLINK NAME=5 SLAVETYP=LINE SLAVENAM=7 MASTERTY=POINT,
     MASTERNA=17 DOFSI=123456
@CLEAR
3
@


***********************************************
***************                 ***************
***************     SUPPORT     ***************
***************   CONDITIONS    ***************
***************                 ***************
***********************************************


FIXITY NAME=BC1
@CLEAR
 'X-TRANSLATION'
 'Y-TRANSLATION'
 'Z-TRANSLATION'
 'X-ROTATION'
@
```

```
FIXITY NAME=BC2
@CLEAR
 'Y-TRANSLATION'
 'Z-TRANSLATION'
 'X-ROTATION'
@

FIXBOUNDARY POINTS
@CLEAR
16  'BC1'
18  'BC2'
@


************************************************
***************                  ***************
***************       LOADS      ***************
***************                  ***************
************************************************



TIMESTEP
@CLEAR
100 1
@

LOAD FORCE NAME=1 MAGNITUD=100000 FX=0 FY=0 FZ=-1

APPLY-LOAD
@CLEAR
1  'FORCE' 1  'POINT' 17
@

IMPERFECTION POINTS
@CLEAR
1 17 2 0.0001
@


************************************************
***************                  ***************
***************     ANALYSIS     ***************
***************                  ***************
************************************************



TOLERANCES ITERATION CONVERGE=EF ETOL=1E-08 RTOL=0.01

MASTER ANALYSIS=STATIC MODEX=EXECUTE,
     TSTART=0.00000000000000 IDOF=0 OVALIZAT=NONE,
     FLUIDPOT=AUTOMATIC CYCLICPA=1 IPOSIT=STOP,
     REACTION=YES INITIALS=NO FSINTERA=NO IRINT=DEFAULT,
     CMASS=NO SHELLNDO=AUTOMATIC AUTOMATI=LDC,
     SOLVER=SPARSE CONTACT-=CONSTRAINT-FUNCTION,
     TRELEASE=0.00000000000000 RESTART-=NO FRACTURE=NO,
     LOAD-CAS=NO LOAD-PEN=NO SINGULAR=YES,
     STIFFNES=0.000100000000000000 MAP-OUTP=NONE,
     MAP-FORM=NO NODAL-DE='' POROUS-C=NO ADAPTIVE=0,
     ZOOM-LAB=1 AXIS-CYC=0 PERIODIC=NO VECTOR-S=GEOMETRY,
     EPSI-FIR=NO STABILIZ=NO STABFACT=1.00000000000000E-10,
     RESULTS=PORTHOLE FEFCORR=NO BOLTSTEP=1 EXTEND-S=YES,
     CONVERT-=NO DEGEN=YES TMC-MODE=NO ENSIGHT-=NO,
     IRSTEPS=1 INITIALT=NO TEMP-INT=NO ESINTERA=NO,
     OP2GEOM=NO INSITU-D=NO OP2ERCS=ELEMENT 2DPL-AX=YZ-Z

AUTOMATIC LOAD-DISPLACEMENT POINT=17 DOF=3,
     DISPLACE=-0.00005 DISPMX=0.15 CONTINUE=YES TYPE=POINT
```

## B.3.2 Uniformly distributed load with an eccentricity of $y_g = 5$ mm

Here follows the two ADINA-IN Command files, one for each type of analysis, used to perform the study of the 2.8 m long channel beam in ADINA, subjected to a uniformly distributed load over the whole span, with an eccentricity of $y_g = 5$ mm. A linearized buckling analysis was performed to obtain the first buckling mode shape, to be used as the initial geometric imperfection in the collapse analysis with the LDC method, as described in Section 4.2.6.

### B.3.2.1 Linearized buckling analysis

```
*************************************************
****                                         ****
****       MODIFIED UPE160 CHANNEL BEAM      ****
****              LENGTH L=2.8M              ****
****       LINEARIZED BUCKLING ANALYSIS      ****
****        UNIFORMLY DISTRIBUTED LOAD       ****
****            ECCENTRICITY YG=5MM          ****
****                                         ****
*************************************************


*************************************************
***************                  ****************
***************    ASSUMPTIONS   ****************
***************                  ****************
*************************************************


FEPROGRAM ADINA

KINEMATICS DISPLACE=LARGE STRAINS=SMALL


*************************************************
***************                  ****************
***************     GEOMETRY     ****************
***************                  ****************
*************************************************


COORDINATES POINT SYSTEM=0
@CLEAR
1  0     0.01929  0.075 0
2  0     0.01929 -0.075 0
3  2.8   0.01929  0.075 0
4  2.8   0.01929 -0.075 0
5  0    -0.04746  0.075 0
6  0    -0.04746 -0.075 0
7  2.8  -0.04746  0.075 0
8  2.8  -0.04746 -0.075 0
9  0     0        0     0
10 2.8   0        0     0
11 0     0.04610  0     0
12 2.8   0.04610  0     0
13 0     0.01929  0     0
14 2.8   0.01929  0     0
15 0     0.04110  0     0
16 2.8   0.04110  0     0
@
```

```
SURFACE VERTEX NAME=1 P1=1 P2=13 P3=14 P4=3
SURFACE VERTEX NAME=2 P1=13 P2=2 P3=4 P4=14
SURFACE VERTEX NAME=3 P1=1 P2=5 P3=7 P4=3
SURFACE VERTEX NAME=4 P1=2 P2=6 P3=8 P4=4

LINE STRAIGHT NAME=14 P1=15 P2=16


************************************************
***************                  ***************
***************   MATERIALS &    ***************
***************    SECTIONS      ***************
***************                  ***************
************************************************


MATERIAL ELASTIC NAME=1 E=2.1E+11 NU=0.3 MDESCRIP='STEEL'

MATERIAL ELASTIC NAME=2 E=1 NU=0 MDESCRIP='Load line'

CROSS-SECTIO PIPE NAME=1 DIAMETER=0.001 SOLID=YES


************************************************
***************                  ***************
***************  ELEMENT GROUPS  ***************
***************                  ***************
************************************************


EGROUP SHELL NAME=1 MATERIAL=1 RINT=2 SINT=2 TINT=7,
     DESCRIPT='WEB' THICKNES=0.0065 TINT-TYP=NEWTON-COTES,
     SHELLTY=STANDARD

EGROUP SHELL NAME=2 MATERIAL=1 RINT=2 SINT=2 TINT=7,
     DESCRIPT='FLANGES' THICKNES=0.01 TINT-TYP=NEWTON-COTES,
     SHELLTY=STANDARD

EGROUP BEAM NAME=3 DISPLACE=LARGE MATERIAL=2,
     DESCRIPT='Load line' SECTION=1


************************************************
***************                  ***************
***************     ELEMENT      ***************
***************   SUBDIVISION    ***************
***************                  ***************
************************************************


SUBDIVIDE SURFACE NAME=1 NDIV1=3 NDIV2=28
@CLEAR
2
@

SUBDIVIDE SURFACE NAME=3 NDIV1=4 NDIV2=28
@CLEAR
4
@

SUBDIVIDE LINE NAME=14 NDIV=28
```

```
GSURFACE NODES=4 NCTOLERA=1E-05 GROUP=1 PREFSHAP=QUADRILATERAL
@CLEAR
1
2
@

GSURFACE NODES=4 NCTOLERA=1E-05 GROUP=2 PREFSHAP=QUADRILATERAL
@CLEAR
3
4
@

GLINE NODES=2 NCTOLERA=1E-05 GROUP=3 XO=0 YO=1 ZO=0
@CLEAR
14
@

GPOINT NODE=465 NCOINCID=ALL NCTOLERA=1E-05
@CLEAR
11
12
@



************************************************
***************                  ***************
***************    RIGID LINKS   ***************
***************                  ***************
************************************************


* Comment: Two separate rigid links for each end (support condition)
*          to allow warping

NODESET NAME=1 DESCRIPT='Web 1'
@CLEAR
2
3
117
118
@

NODESET NAME=2 DESCRIPT='Flanges 1'
@CLEAR
1
204
205
206
207
119
320
321
322
323
@

RIGIDLINK NAME=1 SLAVETYP=NODESET SLAVENAM=1 MASTERTY=POINT,
     MASTERNA=11 DOFSI=123456

RIGIDLINK NAME=2 SLAVETYP=NODESET SLAVENAM=2 MASTERTY=POINT,
     MASTERNA=11 DOFSI=23

NODESET NAME=3 DESCRIPT='Web 2'
@CLEAR
114
115
201
202
@
```

```
NODESET NAME=4 DESCRIPT='Flanges 2'
@CLEAR
113
316
317
318
319
203
432
433
434
435
@

RIGIDLINK NAME=3 SLAVETYP=NODESET SLAVENAM=3 MASTERTY=POINT,
     MASTERNA=12 DOFSI=123456

RIGIDLINK NAME=4 SLAVETYP=NODESET SLAVENAM=4 MASTERTY=POINT,
     MASTERNA=12 DOFSI=23

* Comment: Centre line of web 'master' and line of load
*          application 'slave'

RIGIDLINK NAME=5 SLAVETYP=LINE SLAVENAM=14 MASTERTY=LINE,
     MASTERNA=2 DOFSI=123456


*************************************************
***************                  ***************
***************     SUPPORT      ***************
***************    CONDITIONS    ***************
***************                  ***************
*************************************************


FIXITY NAME=BC1
@CLEAR
 'X-TRANSLATION'
 'Y-TRANSLATION'
 'Z-TRANSLATION'
 'X-ROTATION'
@

FIXITY NAME=BC2
@CLEAR
 'Y-TRANSLATION'
 'Z-TRANSLATION'
 'X-ROTATION'
@

FIXBOUNDARY POINTS
@CLEAR
11  'BC1'
12  'BC2'
@


*************************************************
***************                  ***************
***************      LOADS       ***************
***************                  ***************
*************************************************


LOAD LINE NAME=1 MAGNITUD=-40000
```

```
APPLY-LOAD
@CLEAR
1  'LINE' 1  'LINE' 14 0 1 0 13 -1 7
@


************************************************
***************                  ***************
***************     ANALYSIS     ***************
***************                  ***************
************************************************



TOLERANCES ITERATION CONVERGE=EF ETOL=1E-08 RTOL=0.01

MASTER ANALYSIS=BUCKLING-LOADS MODEX=EXECUTE,
     TSTART=0.00000000000000 IDOF=0 OVALIZAT=NONE,
     FLUIDPOT=AUTOMATIC CYCLICPA=1 IPOSIT=STOP,
     REACTION=YES INITIALS=NO FSINTERA=NO IRINT=DEFAULT,
     CMASS=NO SHELLNDO=AUTOMATIC AUTOMATI=OFF,
     SOLVER=SPARSE CONTACT-=CONSTRAINT-FUNCTION,
     TRELEASE=0.00000000000000 RESTART-=NO FRACTURE=NO,
     LOAD-CAS=NO LOAD-PEN=NO SINGULAR=YES,
     STIFFNES=0.000100000000000000 MAP-OUTP=NONE,
     MAP-FORM=NO NODAL-DE='' POROUS-C=NO ADAPTIVE=0,
     ZOOM-LAB=1 AXIS-CYC=0 PERIODIC=NO VECTOR-S=GEOMETRY,
     EPSI-FIR=NO STABILIZ=NO,
     STABFACT=1.00000000000000E-10 RESULTS=PORTHOLE,
     FEFCORR=NO BOLTSTEP=1 EXTEND-S=YES CONVERT-=NO,
     DEGEN=YES TMC-MODE=NO ENSIGHT-=NO IRSTEPS=1,
     INITIALT=NO TEMP-INT=NO ESINTERA=NO OP2GEOM=NO,
     INSITU-D=NO OP2ERCS=ELEMENT 2DPL-AX=YZ-Z

BUCKLING-LOA NEIGEN=1 METHOD=CLASSICAL
```

#### B.3.2.2 Collapse analysis with the LDC method

```
************************************************
****                                        ****
****       MODIFIED UPE160 CHANNEL BEAM     ****
****              LENGTH L=2.8M             ****
****  COLLAPSE ANALYSIS WITH THE LDC METHOD ****
****        UNIFORMLY DISTRIBUTED LOAD      ****
****            ECCENTRICITY YG=5MM         ****
****                                        ****
************************************************


************************************************
***************                  ***************
***************    ASSUMPTIONS   ***************
***************                  ***************
************************************************



FEPROGRAM ADINA

KINEMATICS DISPLACE=LARGE STRAINS=SMALL


************************************************
***************                  ***************
***************     GEOMETRY     ***************
***************                  ***************
************************************************
```

```
COORDINATES POINT SYSTEM=0
@CLEAR
1  0    0.01929  0.075 0
2  0    0.01929 -0.075 0
3  2.8  0.01929  0.075 0
4  2.8  0.01929 -0.075 0
5  0   -0.04746  0.075 0
6  0   -0.04746 -0.075 0
7  2.8 -0.04746  0.075 0
8  2.8 -0.04746 -0.075 0
9  0    0        0     0
10 2.8  0        0     0
11 0    0.04610  0     0
12 2.8  0.04610  0     0
13 0    0.01929  0     0
14 2.8  0.01929  0     0
15 0    0.04110  0     0
16 2.8  0.04110  0     0
@

SURFACE VERTEX NAME=1 P1=1 P2=13 P3=14 P4=3
SURFACE VERTEX NAME=2 P1=13 P2=2 P3=4 P4=14
SURFACE VERTEX NAME=3 P1=1 P2=5 P3=7 P4=3
SURFACE VERTEX NAME=4 P1=2 P2=6 P3=8 P4=4

LINE STRAIGHT NAME=14 P1=15 P2=16


************************************************
***************                  ***************
***************    MATERIALS &   ***************
***************     SECTIONS     ***************
***************                  ***************
************************************************


MATERIAL ELASTIC NAME=1 E=2.1E+11 NU=0.3 MDESCRIP='STEEL'

MATERIAL ELASTIC NAME=2 E=1 NU=0 MDESCRIP='Load line'

CROSS-SECTIO PIPE NAME=1 DIAMETER=0.001 SOLID=YES


************************************************
***************                  ***************
***************  ELEMENT GROUPS  ***************
***************                  ***************
************************************************


EGROUP SHELL NAME=1 MATERIAL=1 RINT=2 SINT=2 TINT=7,
     DESCRIPT='WEB' THICKNES=0.0065 TINT-TYP=NEWTON-COTES,
     SHELLTY=STANDARD

EGROUP SHELL NAME=2 MATERIAL=1 RINT=2 SINT=2 TINT=7,
     DESCRIPT='FLANGES' THICKNES=0.01 TINT-TYP=NEWTON-COTES,
     SHELLTY=STANDARD

EGROUP BEAM NAME=3 DISPLACE=LARGE MATERIAL=2,
     DESCRIPT='Load line' SECTION=1
```

```
************************************************
***************                  ***************
***************     ELEMENT      ***************
***************   SUBDIVISION    ***************
***************                  ***************
************************************************



SUBDIVIDE SURFACE NAME=1 NDIV1=3 NDIV2=28
@CLEAR
2
@

SUBDIVIDE SURFACE NAME=3 NDIV1=4 NDIV2=28
@CLEAR
4
@

SUBDIVIDE LINE NAME=14 NDIV=28

GSURFACE NODES=4 NCTOLERA=1E-05 GROUP=1 PREFSHAP=QUADRILATERAL
@CLEAR
1
2
@

GSURFACE NODES=4 NCTOLERA=1E-05 GROUP=2 PREFSHAP=QUADRILATERAL
@CLEAR
3
4
@

GLINE NODES=2 NCTOLERA=1E-05 GROUP=3 XO=0 YO=1 ZO=0
@CLEAR
14
@

GPOINT NODE=465 NCOINCID=ALL NCTOLERA=1E-05
@CLEAR
11
12
@



************************************************
***************                  ***************
***************   RIGID LINKS    ***************
***************                  ***************
************************************************



* Comment: Two separate rigid links for each end (support condition)
*          to allow warping

NODESET NAME=1 DESCRIPT='Web 1'
@CLEAR
2
3
117
118
@
```

```
NODESET NAME=2 DESCRIPT='Flanges 1'
@CLEAR
1
204
205
206
207
119
320
321
322
323
@

RIGIDLINK NAME=1 SLAVETYP=NODESET SLAVENAM=1 MASTERTY=POINT,
     MASTERNA=11 DOFSI=123456

RIGIDLINK NAME=2 SLAVETYP=NODESET SLAVENAM=2 MASTERTY=POINT,
     MASTERNA=11 DOFSI=23

NODESET NAME=3 DESCRIPT='Web 2'
@CLEAR
114
115
201
202
@

NODESET NAME=4 DESCRIPT='Flanges 2'
@CLEAR
113
316
317
318
319
203
432
433
434
435
@

RIGIDLINK NAME=3 SLAVETYP=NODESET SLAVENAM=3 MASTERTY=POINT,
     MASTERNA=12 DOFSI=123456

RIGIDLINK NAME=4 SLAVETYP=NODESET SLAVENAM=4 MASTERTY=POINT,
     MASTERNA=12 DOFSI=23

* Comment: Centre line of web 'master' and line of load
*          application 'slave'

RIGIDLINK NAME=5 SLAVETYP=LINE SLAVENAM=14 MASTERTY=LINE,
     MASTERNA=2 DOFSI=123456


************************************************
***************                  ***************
***************     SUPPORT      ***************
***************    CONDITIONS    ***************
***************                  ***************
************************************************


FIXITY NAME=BC1
@CLEAR
 'X-TRANSLATION'
 'Y-TRANSLATION'
 'Z-TRANSLATION'
 'X-ROTATION'
@
```

```
FIXITY NAME=BC2
@CLEAR
 'Y-TRANSLATION'
 'Z-TRANSLATION'
 'X-ROTATION'
@

FIXBOUNDARY POINTS
@CLEAR
11  'BC1'
12  'BC2'
@


*************************************************
***************                  ***************
***************       LOADS      ***************
***************                  ***************
*************************************************


TIMESTEP NAME=DEFAULT
@CLEAR
100 1
@

LOAD LINE NAME=1 MAGNITUD=-100000

APPLY-LOAD
@CLEAR
1  'LINE' 1  'LINE' 14 0 1 0 13 -1 7
@

IMPERFECTION NODES
@CLEAR
1 60 2 -0.0001
@


*************************************************
***************                  ***************
***************     ANALYSIS     ***************
***************                  ***************
*************************************************


TOLERANCES ITERATION CONVERGE=EF ETOL=1E-08 RTOL=0.01

MASTER ANALYSIS=STATIC MODEX=EXECUTE,
     TSTART=0.00000000000000 IDOF=0 OVALIZAT=NONE,
     FLUIDPOT=AUTOMATIC CYCLICPA=1 IPOSIT=STOP,
     REACTION=YES INITIALS=NO FSINTERA=NO IRINT=DEFAULT,
     CMASS=NO SHELLNDO=AUTOMATIC AUTOMATI=LDC,
     SOLVER=SPARSE CONTACT-=CONSTRAINT-FUNCTION,
     TRELEASE=0.00000000000000 RESTART-=NO FRACTURE=NO,
     LOAD-CAS=NO LOAD-PEN=NO SINGULAR=YES,
     STIFFNES=0.000100000000000000 MAP-OUTP=NONE,
     MAP-FORM=NO NODAL-DE='' POROUS-C=NO ADAPTIVE=0,
     ZOOM-LAB=1 AXIS-CYC=0 PERIODIC=NO VECTOR-S=GEOMETRY,
     EPSI-FIR=NO STABILIZ=NO STABFACT=1.00000000000000E-10,
     RESULTS=PORTHOLE FEFCORR=NO BOLTSTEP=1 EXTEND-S=YES,
     CONVERT-=NO DEGEN=YES TMC-MODE=NO ENSIGHT-=NO,
     IRSTEPS=1 INITIALT=NO TEMP-INT=NO ESINTERA=NO,
     OP2GEOM=NO INSITU-D=NO OP2ERCS=ELEMENT 2DPL-AX=YZ-Z

AUTOMATIC LOAD-DISPLACEMENT NODE=60 DOF=3,
     DISPLACE=-0.00005 DISPMX=0.15 CONTINUE=YES TYPE=NODE
```