BUCKLING OF A COLUMN UNDER A COMPRESSIVE AXIAL LOAD

Palkin nurjahdus aksiaalisessa puristavassa kuormituksessa

Lappeenrannan–Lahden teknillinen yliopisto LUT

Konetekniikan kandidaatintyö

2022

Klaus Kuhanen

Tarkastaja: TkT Babak Bozorgmehri

TIIVISTELMÄ

Lappeenrannan–Lahden teknillinen yliopisto LUT

Oma schoolisi: LUT Teknis-luonnontieteellinen

Oma koulutusohjelmasi: Konetekniikka

Klaus Kuhanen

Palkin nurjahdus aksiaalisessa puristavassa kuormituksessa

Koulutusohjelmasi ja tutkielmasi: Konetekniikan kandidaatintyö

2022

31 sivua, 8 kuvaa, 4 taulukkoa ja 2 liitettä

Tarkastaja: TkT Babak Bozorgmehri

Avainsanat: Nurjahdus, elementtimenetelmä, FEMAP, MATLAB, kriittinen kuormitus, siirtymä

Tässä kandidaatintyössä palkin nurjahdusta simuloitiin elementtimenetelmää hyväksi käyttäen. Työssä käytettiin FE-ohjelmisto FEMAP:a sekä MATLAB-ohjelmistoa. Työn tavoitteena oli löytää paras ratkaisu menetelmä nurjahdus ongelmien ratkaisemiseen. Tällöin voidaan välttyä suurilta taloudellisilta kustannuksilta sekä henkilöstö vahingoilta palkin nurjahduksen tapahtuessa. Nurjahdus johtuu kriittisen kuormituksen ylittävästä puristavasta aksiaalisesta kuormituksesta. Kun palkki nurjahtaa se menettää vakautensa ja siitä tulee epävakaa. Tällöin palkki ei enää voi tukea tarvittavaa kuormaa.

FEMAP:a ja MATLAB:a käytetään ratkaisemaan tiettyjä nurjahdus ongelmia. Ratkaistavat ongelmat sisältävät teräsrakenteen, jossa palkki nivelöidyillä päillä sekä rautatie raiteen, joka on jäykästi kiinnitetty molemmista päistä. FEMAP:lla ja MATLAB:lla ei voida ratkaista täysin samanlaista ongelmaa, joten ongelmissa on eroja ohjelmistosta riippuen. FEMAP:n nurjahdus analyysistä saatiin ulos monia arvoja kuten jännityksiä, solmusiirtymiä, ominaisarvoja sekä kriittisiä kuormituksia. FEMAP:ssa palkkien profiilit olivat monimutkaisempia ja lähempänä reaalielämän tilannetta varsinkin rautatie raiteen tapauksessa. FEMAP:lla pystyttiin luomaan käytännössä samanlainen profiili kuin oikealla raiteella kun taas MATLAB:lla jouduttiin tyytymään neliö palkki profiiliin. Monimutkaisen profiilin teko olisi onnistunut MATLAB:lla, mutta se olisi monimutkaistanut koodia huomattavasti työn laajuuden ulkopuolelle. MATLAB:sta saatiin ulos palkin solmusiirtymät lineaaristaattisessa tapauksessa, joka simuloi palkin käyttäytymistä varsin hyvin. MATLAB:lla olisi täytynyt

tehdä ominaisarvo taajuus analyysi, jotta olisi saatu palkin ominaisarvo ja täten kriittinen kuormitus. Tämä olisi taas tehnyt MATLAB koodista liian monimutkaisen.

FEMAP oli nurjahduksen kannalta kattavampi ratkaisu metodi kuin MATLAB. FEMAP:n nurjahdus analyysi oli metodeista visuaalisempi, sillä se esitti palkin 3D-mallin ja sen muodonmuutoksen. MATLAB:lla voitiin taulukoida palkin solmusiirtymät havainnollistamaan siirtymiä paremmin vaikkei siltikään yhtä hyvin kuin FEMAP:ssa.

ABSTRACT

Lappeenranta–Lahti University of Technology LUT

Your school: School of Engineering Science

Mechanical Engineering

Klaus Kuhanen

Buckling of a column under a compressive axial load

Bachelor's thesis

2022

31 pages, 8 figures, 4 tables and 2 appendices

Examiner: D.Sc. Babak Bozorgmehri

Keywords: Buckling, finite element method, FEMAP, MATLAB, critical load, displacement

In this thesis, the buckling of columns under compressive axial loads was simulated with finite element methods. Finite element methods and analytical solution method were used. The finite element methods include commercial finite element code FEMAP and in-house academic code MATLAB. The objective was to simulate buckling problems so large financial expenses and other collateral damages could be avoided in case of buckling. Generally, buckling is a result of instability in a column. This instability is usually caused by a compressive axial load that is larger than the columns critical load.

FEMAP and MATLAB were used to solve certain buckling problems using finite element methods. The problems included a steel structure that had a column with pinned boundaty conditions in both ends and a railway track that had fixed boundary conditions in both ends. Identical problems could not be solved with both the softwares, so, there were some differences in the problems solved with MATLAB and FEMAP. From FEMAP buckling analysis a lot of different values were obtained such as stresses, nodal displacements, eigenvalues and critical loads. In FEMAP the profile of the column was more complex and was closer to the real-life situation especially in the railway track problem. The profile of the railway track

rail was almost the same in FEMAP as it is in real-life. In MATLAB a square column profile had to be used so the code wouldn't get too complicated. Nodal displacements were obtained from the MATLAB code which simulate the behavior of the column well.

As a solution method for a buckling problem FEMAP turned out to be more extent than MATLAB. FEMAPs buckling analysis was more visually helpful as it would show the columns transformation with the help of a 3D-model. In MATLAB a graph can be created to show the columns transformation more clearly.

LIST OF SYMBOLS AND ABBREVIATIONS

Roman

F	Force	[N]
E	Modulus of elasticity	[Pa]
I	Moment of inertia	[m^4]
L	Effective length	[m]
A	Cross-sectional area	[m^2]

Greek

σ	Axial stress	[Pa]

Superscripts and subscripts

$(.)_{cr}$	Critical
$(.)_{a}$	Applied load
$(.)_{i}$	Eigenvalue index number
$(.)_{d}$	Differential

Abbreviation

3D	Three dimensional
FE	Finite element

Table of content

List of tables

1 Introduction

In many modern structures and machines there are beam-like structures under compressive axial load. A column can buckle locally, or the buckling can happen in the whole column. It is vital to examine what can cause buckling failures in these columns so we can avoid big economical expenses and more importantly we will be able to avoid human casualties. If an important column loses its stability, it can collapse the entire system. This kind of disaster can bankrupt a company. We are trying to simulate the buckling phenomena so we can predict failure and be prepared for that. When we can predict failure, we can come up a solution for it.

This Bachelor's thesis is about the buckling of a column under compressive axial load. In this introduction we are going through the reasons for this study and what we want to accomplish. We also do a brief literature review of past studies and explain the course of the rest of the study.

1.1 Background to the study

Euler's formula is an important analytical formula when calculating buckling of a column (Beer et al. 2012, 636). Euler's formula is used to calculate the critical load that leads to buckling. The critical load is the threshold of the allowed load applied under which buckling starts. If the applied load in the column is larger than the critical load, then the column loses its stability because of buckling. The Euler's formula consists of different variables such as modulus of elasticity, moment of inertia and effective length. The modulus of elasticity depends on the material of the column, the moment of inertia depends on the dimensions of the column and finally the effective length is dependent on the length of the column as well as the boundary conditions of the column. The boundary conditions can be for example fixed or pinned end. Critical stress can be calculated when critical load is divided by the cross-sectional area of the column. (Beer et al. 2012, 632-637.)

Many of the previous studies about buckling of a column are focused on a certain column structure for example high strength steel welded box-section columns (Li et al. 2022). Previous simple solution for buckling has been adding supports to a column so its effective length decreases and therefore the critical load increases (Bedford et al. 2020). Short steel columns have been repaired post buckling but our focus in this study is in solving buckling problems by using different approaches to find out the best solution procedure to the real-life buckling problems. (Kaya et al. 2015). We are focusing more generally on buckling problems for which we want to simulate a computational model as a solution procedure. We also try to solve real life buckling problems such as buckling of a railway track.

1.2 Research methods and goal of the study

Goal of the study is to find solutions to buckling by simulating it so failures can be prevented in the future. Three different approaches are used to solve buckling problem and find out which method suits best to solving buckling problems. These methods are analytical solution that mainly consists of Euler's formula and finite element solutions with (a) an academic code- MATLAB (b) commercial software (FEMAP). Therefore, three different kinds of solutions can be obtained which can be analyzed and compared. Then the best solution method is chosen for certain real-life buckling problem. This study is only focusing on compressive axial load, so moment or combination of moment and axial load are not the focus (Figure 1). In Figure 1, P is axial load and M is bending moment.

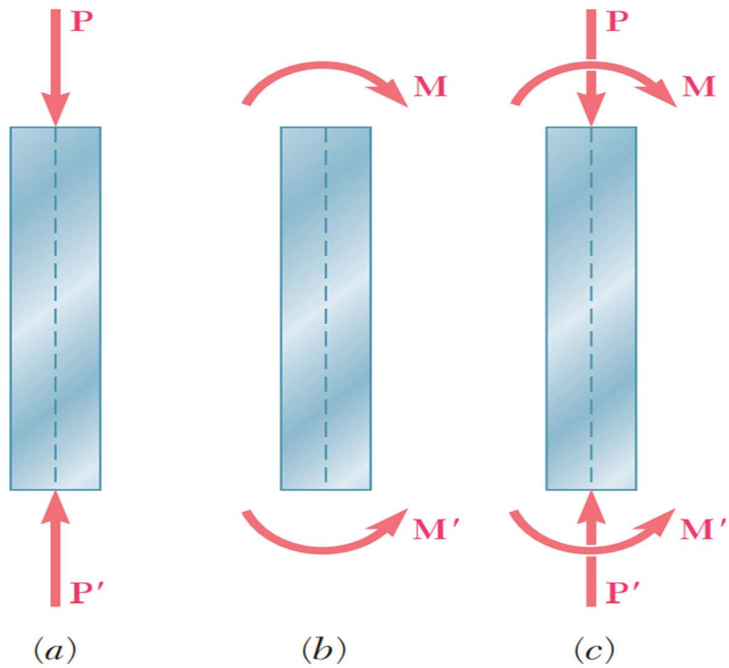


Figure 1. Different loads affecting columns (Mechanics of materials, 2012, pp. 676).

In the following chapters we go through the methods used in this study in details. After going through the working principles of these three different methods, we use these methods in real-life buckling problems. We calculate the buckling in a simple steel structure and a railway track rail that are both under compressive axial load. Finally, the results are analyzed and compared to each other.

2 Buckling research methods

In this chapter we are explaining the methods to this study. We go through the principles of how finite element methods are used in FEMAP and Matlab. This gives us insight on how these softwares help us solve different buckling problems. We also explain the analytical solution that can be used if no softwares are available to help with calculations.

2.1 Analytical solution method

Analytical solution to buckling is acquired by using the Euler's formula shown in Eq. (2.1). From Euler's equation we get the critical load that causes the column to buckle and there for lose its stability. many different things affect the critical load in a column such as the columns profile, material and length. The modulus of elasticity E comes from the material of the column, moment of inertia I comes from the profile of the column and lastly the L is the effective length of the column. (Beer et al. 2012, 635-636.)

$$P_{cr} = \frac{\pi^2 EI}{L^2} \tag{2.1}$$

The effective length is different than the whole length of the column. The columns boundary conditions have an effect on the effective length. From Figure 2 we can see how the boundary conditions affect the effective length. When the column is fixed from one end and free on the other the effective length is twice the length of the column. If both ends are pinned the effective length is the length of the column. On a column that has one fixed end and one pinned end, the effective length is 0,7 times the length of the column. And if both ends are fixed it's 0,5 times the length of the column. As can be seen from Eq (2.1) the effective length L is squared so it has a lot of impact to the critical load. If the effective length doubles the critical load goes down to a quarter of what it was. (Beer et al. 2012, 642.)

$$\sigma_{cr} = \frac{P_{cr}}{A} \tag{2.2}$$

After we have the critical load, we can calculate the critical stress by using the Eq. (2.2) where A is the cross-sectional area of the column. (Tekniikan kaavasto 2021, 142.)

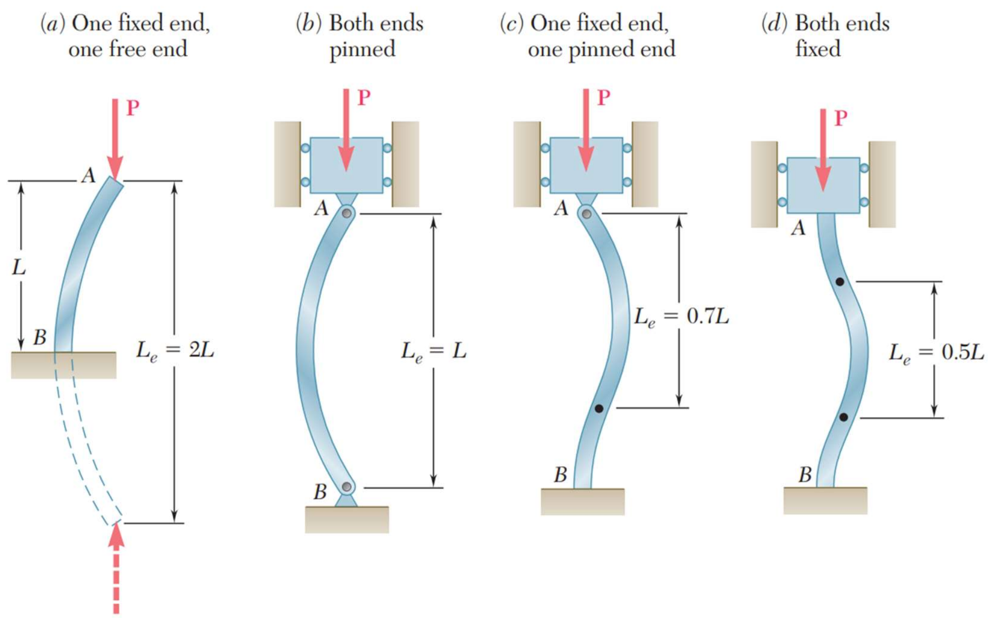


Figure 2. Columns boundary conditions effect on effective length (Mechanics of materials, 2012, pp.642)

In this study we are going to have closer look at real-life buckling problems that include columns that have pinned-pinned boundary conditions and fixed-fixed boundary conditions.

2.2 Finite element methods–In-house academic code

Finite element code is able to solve uniaxially loaded problem with finite element code by using one dimensional two node linear rod elements. MATLAB is used to run this code. Some geometrical and material related variables need to be inserted such as modulus of elasticity and cross-sectional measurements for example. In this study only linear beam elements are considered. Beam element based on the Euler-Bernoulli beam theory was used in the MATLAB code. The polynomial for the quadratic element is shown in Eq. (2.3).

$$\mathbf{p} = [1, x, x^2, x^3] \tag{2.3}$$

These is the monomial for the shape function vectors. With this monomial the shape function vector for axial can be formed. Eq. (2.4) is the shape function vector to describe bending and Eq. (2.5) is the shape function vector to describe axial deformation. *L* is the length of the structure.

$$\mathbf{N}_v{}^{[y]} = [1 - \frac{x}{L}, \frac{x}{L}] \tag{2.4}$$

$$\mathbf{N}_v{}^{[x]} = [(\frac{2x^2}{L^2} - \frac{3x}{L} + 1, \frac{x}{L}, \frac{4x}{L} - \frac{4x^2}{L^2}, \frac{2x^2}{L^2} - \frac{x}{L}] \tag{2.5}$$

Consistent mass matrix can be formed via the shape functions. Mass matrix is required for eigenvalue analysis that is needed to calculate the critical load when using MATLAB finite element code. Next the element force vector (Eq. 2.6) and element nodal displacement vector (Eq. 2.7) can be written.

$$\mathbf{f}^{(e)} = \begin{pmatrix} \mathrm{f}_1^{(e)} \\ \mathrm{f}_2^{(e)} \end{pmatrix} \tag{2.6}$$

$$\mathbf{u}^{e} = \begin{pmatrix} \mathrm{u}_1^{(e)} \\ \mathrm{u}_2^{(e)} \end{pmatrix} \tag{2.7}$$

After this the stiffness matrices for linear FE elements can be directly computed. First the element matrices are created and then they are combined to create the global matrix. Element matrix is four by four matrix as shown in Eq. (2.8). Elements 1 and 2 are interconnected via nodal displacement meaning that first elements second node has the same nodal displacement as the second elements first node. When the two element matrices are combined, they create three by three global matrix (Eq. 2.9). The superscript indicates which matrix is for which element.

$$\mathbf{K}_1{}^{(1)} = \begin{pmatrix} K_{11}{}^{(1)} & K_{12}{}^{(1)} \\ K_{21}{}^{(1)} & K_{22}{}^{(1)} \end{pmatrix} \tag{2.8}$$

$$\mathbf{K} = \begin{pmatrix} K_{11}{}^{(1)} & K_{12}{}^{(1)} & 0 \\ K_{21}{}^{(1)} & K_{22}{}^{(1)} + K_{11}{}^{(2)} & K_{12}{}^{(2)} \\ 0 & K_{21}{}^{(2)} & K_{22}{}^{(2)} \end{pmatrix} \tag{2.9}$$

In the FE model the stiffness matrix is always constant and does not depend on the coordinates. So, when the axial force is also constant in the FE model, the equilibrium equation remains almost constant with using more elements.

$$\begin{pmatrix} \mathrm{u}_1 \\ \mathrm{u}_2 \\ \mathrm{u}_3 \end{pmatrix} = \begin{pmatrix} K_{11}{}^{(1)} & K_{12}{}^{(1)} & 0 \\ K_{21}{}^{(1)} & K_{22}{}^{(1)} + K_{11}{}^{(2)} & K_{12}{}^{(2)} \\ 0 & K_{21}{}^{(2)} & K_{22}{}^{(2)} \end{pmatrix}^{-1} \begin{pmatrix} 0 \\ 0 \\ \mathrm{f}_x \end{pmatrix} \tag{2.10}$$

From Eq. (2.10) the nodal displacements can be solved. If there are more elements in the FE model the size of vectors and matrices grow but the solution method still stays the same. These nodal displacements are from a linear static analysis. Eigenvalue frequency analysis is required from MATLAB for eigenvalue and there for the critical load to be obtained.

However, the eigenvalue frequency analysis in MATLAB will not be focused on in this study.

2.3 Finite element methods–Commercial code

Structure is assumed to be in a state of stable equilibrium in Femap's linear static analysis. This means that when the straining force is removed it is assumed that the structure returns to its undeformed position. However, in case of buckling, in under certain combinations of loadings the structure continues to deform instead of returning to its original form even though there is no increase in the magnitude of loading. when this happens, the structure becomes unstable as it buckles. Two key assumptions must be made for linear elastic buckling. There can be no yielding of the structure and that the direction of the forces affecting the structure does not change. (FEMAP 2021.)

Linear elastic buckling includes the effect of the differential stiffness, that includes higher-order strain displacement relationships that are functions of the applied loads, element type and geometry. The differential stiffness represents a linear approximation of softening meaning reducing the stiffness matrix for a compressive axial load and stiffening on the other hand meaning increasing the stiffness matrix for a tensile axial load. (FEMAP 2021.)

In buckling analysis Femap solves the eigenvalue or eigenvalues that are scaling factor which multiply the load applied for the structure to give us the critical buckling load. The solving of the eigenvalues starts with the linear static analysis equation which is presented in Eq. (2.11). (FEMAP 2021.)

$$[K]\{u\} = \{P\} \tag{2.11}$$

K = stiffness matrix which is automatically generated by Simcenter FEMAP with Nastran, based on inserted properties, materials and element geometry, u = the vector of displacements and P is the vector of applied forces on the system. Next the differential stiffness is included and λ is set as an arbitrary scalar multiplier. This equation is the buckling equation (Eq. 2.12). (FEMAP 2021.)

$$([K_a] + \lambda[K_d])\{u\} = [\lambda P_a] \tag{2.12}$$

Here K_a is the systems stiffness matrix, K_d is the differential stiffness matrix and P_a is the applied load on the system. Differential stiffness matrix is a matrix that is necessary to account for the change in potential energy associated with rotation of continuum elements under load. Its properties include that it is symmetric, it is independent of elastic properties, and it depends on element geometry, displacement field, and the state of stress (Femci Book 1995). Now Eq (2.12) can be written as an eigenvalue problem shown in Eq. (2.13). ϕ is the eigenvector of displacement. (FEMAP 2021.)

$$([K_a] - \lambda[K_d])\{\phi\} = 0 \tag{2.13}$$

From Eq. (2.13) we get a condition that the determinate of the matrix must be zero. (Eq. 2.14) The determinate is the part of the equation that is inside | |. This way we can obtain a non-trivial solution. (FEMAP 2021.)

$$|[K_a] - \lambda[K_d]| = 0 \tag{2.14}$$

Now in this eigenvalue problem the values of λ that fulfills the Eq. (2.14) are the critical buckling loads. This eigenproblem can be solved by using the available eigensolvers in

Simcenter Nastran to find the eigenvalues that are the values that multiply the applied load to get the critical buckling load. (FEMAP 2021.)

$$P_{cr,i} = \lambda_i P_a \tag{2.15}$$

In Eq. (2.15), $P_{cr,i}$ is the critical buckling load, λ_i is the eigenvalue given by the solver in FEMAP and P_a is the applied load on the system. Usually only the first eigenvalue is of interest to us. (FEMAP 2021.)

There are multiple eigenvalue extraction methods available in Simcenter Nastran but none of them are perfect for all models. The Lanczos method is the default method because it gives most accuracy for the least cost and there for is the best overall method. We do not go over the methods more closely because they are not the focus of this study. (FEMAP 2021.)

When selecting buckling analysis in FEMAP it requires creation of two subcases which are one that defines the applied static loading and another that defines the eigenvalue solution. Femap creates the eigenvalue subcase automatically when the buckling is set as analysis type. (FEMAP 2021.)

3 Buckling in structures under bi-axial load

In modern structures, mechanics and machinery there are a lot of column parts under compressive axial load. These columns can vary from a simple supportive column to a small part in a high-end cutting-edge machine. Because buckling can happen in almost every column that is under compressive axial load it is important to understand how we can use the theory of buckling to solve real-life problems.

3.1 Solved problems

Next are solving two buckling problems by using two methods mentioned earlier. First, we solve the problems with FEMAP software and the using Matlab code. The cases solved with Femap and Matlab will not be exactly the same because the code in Matlab would be too complicated. These differences are analyzed and discussed more in the conclusions.

3.1.1 Concrete application: Steel compression structure

The first problem that is going to be solved is a steel structure consisting of two beams connected to a wall with a pinned ends and pin connected to each other. Main thing we wanted to solve was the critical load affecting the lower bar which would lead to buckling. The steel structure is presented more accurately later. We started solving this problem by first modelling the structure in FEMAP. The geometry was done by inserting two lines that were connected from their ends. The lines were made by giving the starting and ending coordinates. First coordinates were 0,0,0 and 2000,0,0. This gave us the horizontal line. Second coordinates that gave us the tilted line were 2000,0,0 and 0, -1000,0. Lengths of the beams can be seen in Table 1. The structure can be seen in Figure 3.

After this we made element mesh to these lines. The upper horizontal line had 40 elements and the lower tilted line had 45 elements (Figure 3). Next, we merged the two nodes in the beams point of contact. Both beams are pinned to a wall from the left side. This means in this model only rotation along z-axis is allowed and everything else is prevented. This includes all transition and rotation along x- and y-axis. In the crossing point of the beams, they were pinned to each other. This way the beams could rotate freely along z-axis and move in x- and y-direction connected to each other at the crossing point.

A 10 N/mm load was set on the upper beam. The direction of the load was in the negative y-direction. This load creates compressive axial load to the lower beam which can lead to buckling. Body loads were not considered in this problem. The constraints and loads are visible in Figure 3.

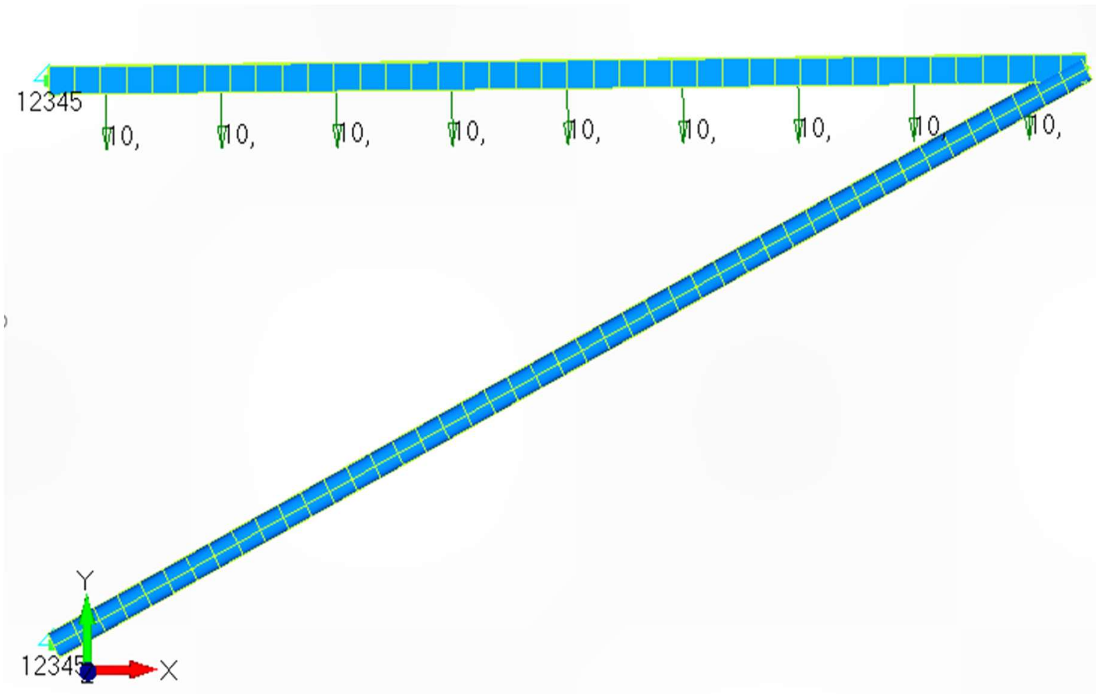
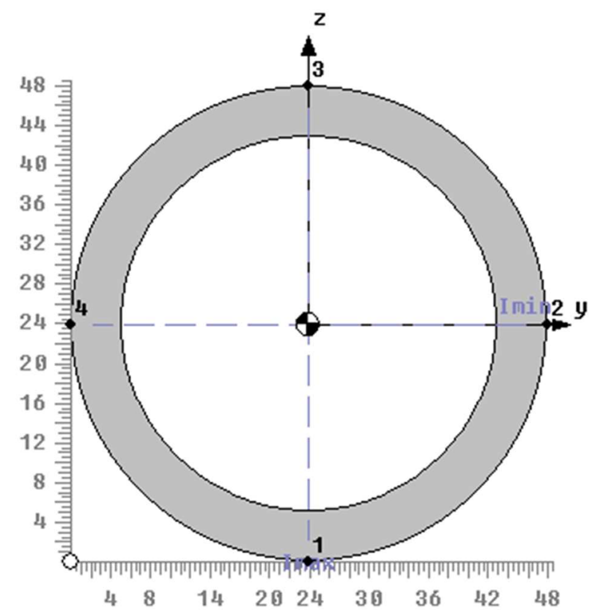


Figure 3. Undeformed FE-model where element mesh, loads and constraints visible.

Beam elements were used on both columns. The upper beam was rectangular bar with measurements of 50 x 30mm, and the lower beam had tube profile with outer diameter 48mm and wall thickness 5mm. The profiles of both beams can be seen in Figure 4. Relevant data on both beams have been gathered in Table 1. For material, steel was used and for that we inserted values for modulus of elasticity and poisson's ratio (Table 2). We also inserted density which is needed in the eigenfrequency analysis.



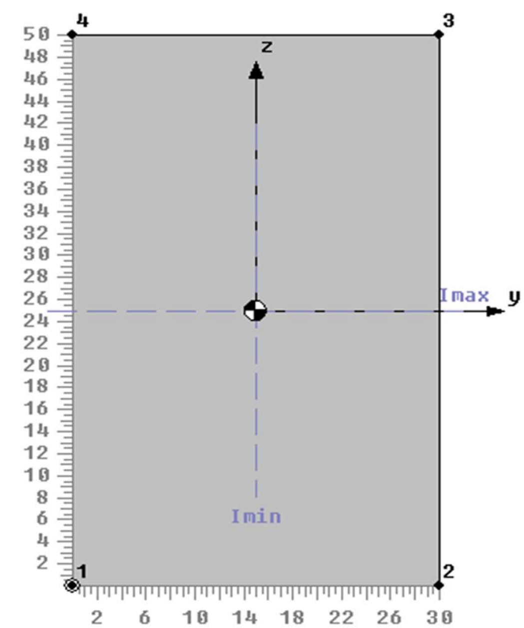


Figure 4. Profiles of the beams

Table 1. Beams profile values

Variable, Unit (Rectangular bar)	**Value**
Height, mm	50
Width, mm	30
Area, mm^2	1500
Moment of inertia I_{zz}, mm^4	112500
Moment of inertia I_{yy}, mm^4	312500
Variable, Unit (Tube)	**Value**
Radius, mm	24
Wall thickness, mm	5
Area, mm^2	675.4424
Moment of inertia I_{zz}, mm^4	158222.4
Moment of inertia I_{yy}, mm^4	158222.4

Buckling analysis was done for the modeled steel structure by using the constraints mentioned before, load and properties. We were looking for the first eigenvalue from which the lower beam would buckle and lose its stability. When the buckling analysis is performed two outputs are received. One for the buckling and another for static case. We are focusing only on the buckling output. From Figure 5 we can see the total translation of the lower beam because of the load affecting it. The buckling deformation is exaggerated in the figure for visualization. The colours on the buckled beam indicate the deformation of the beam. The values for the colours are shown on the right side of the Figure 5 in millimeters. Red indicates the largest deformation in the figure.

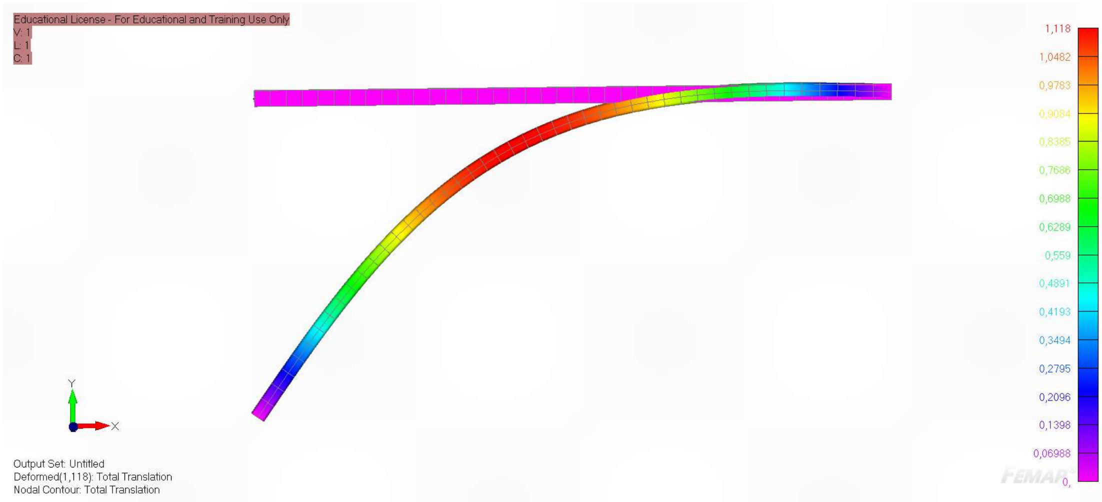


Figure 5. Deformed FE-model

Table 2. Summary table

Variable, unit	**Value**
Upper beam length, mm	**2000**
Lower beam length, mm	**2236.068**
Force affecting the upper beam, N/mm	**-10**
Compressive axial load affecting the lower bar, N	**-22361**
Modulus of elasticity, MPa	**2.1e5**
Poisson’s ratio	**0,3**
Density, Ton/mm^3	**7.85e-9**
Maximum deformation of the lower bar, mm	**1.118**
Maximum stress in the lower beam, MPa	**11.103**

Critical load factor n_{cr}	**2.926642**
Critical load P_{cr}, N	**65442**

The axial compressive force affecting the lower beam is -22361 N. The force is negative because the force is compressive force. From the buckling analysis with FEMAP software we get the critical load factor also known as eigenvalue. When the compressive force is multiplied by the critical load factor the critical load which the beam can withstand can be calculated. The critical load factor needs to be more than one for the beam to not have buckled under the applied load. If it is less than one the critical load is also less than the applied load on the structure. Maximum stress and maximum deformation in the buckled beam were also received from the buckling analysis. These values are presented in Table 2.

3.1.2 Real-life application: Railroad tracks

Buckling of a railway track is the next real-life buckling problem that we are going to solve. The buckling is caused by the force resulting from heat expansion of the rails. Because the rails are fastened at certain points the expansion of the rail causes compressive axial load to the rail which can lead to buckling if the compressive force is large enough. We started solving the real-life railroad track buckling problem by modeling the profile of the track with Solidworks software. Solidworks is 3D-modeling software. We modeled half of the profile and the mirrored it about the center line so its symmetric. The profile of the railroad track was simplified a bit so that it is easier to analyze with femap. The profile of the modeled rail is shown in Figure 6. More detailed measurements for the rail profile can be found in Appendix 1.

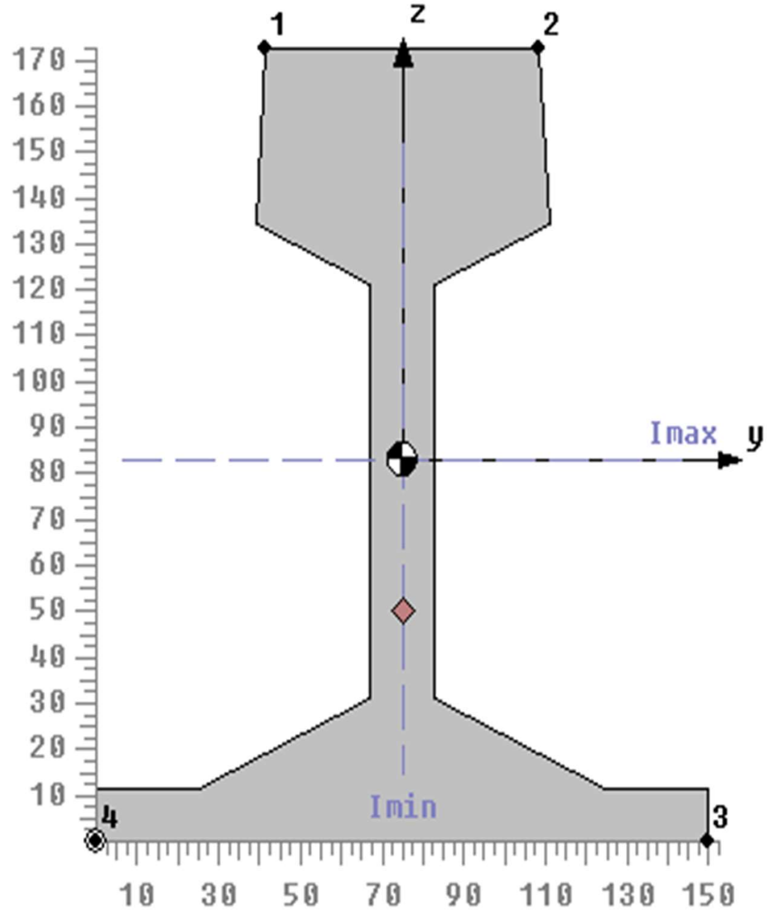


Figure 6. Railway rail profile

Real-life UIC 60 rail profile was used as a template for the modeled rail profile. Height and width of the rail profile are the same as in the UIC 60 rail profile. In Table 3 are gathered values of the modeled rail profile and the UIC 60 profile to compare. As we can see the cross-sectional area and the moments of inertia are relatively same. More measurements and properties of the UIC 60 profile can be found in Appendix 2. As a material we used railway steel which properties such as elastic modulus are also shown in Table 3.

Table 3. Rails profile and material values

Variable, Unit (Modeled profile)	**Value**
Height, mm	172
Width, mm	150
Length, mm	25000
Area, mm^2	7571,754
Moment of inertia I_{zz}, mm^4	5024203
Moment of inertia I_{yy}, mm^4	30785905
Variable, Unit (UIC 60 profile)	**Value**
Height, mm	172
Width, mm	150
Area, mm^2	7670
Moment of inertia I_{zz}, mm^4	5123000
Moment of inertia I_{yy}, mm^4	30383000
Properties, Unit	**Value**
Elastic modulus, MPa	2,07e5
Poisson’s ratio	0,3
Density, ton/mm^3	7,85e-9

The compressive force in the rail is caused by the heat expansion in the rail. The temperature rises 15 degrees of Celsius. In Femap we set the reference temperature in 10 degrees and set the temperature to which it rises. In our case to 25 degrees Celsius. Femap then calculates the loads and stresses that are created in the rail with the help of the coefficient of thermal expansion which is presented in Table 3.

Next, we set up the constraints for the rail. First, we prevent all translation in y-direction because the rail is on the ground and there for cannot move up and down. As for rail beams end constraints both ends should be fixed. This means that all translation and rotation is prevented. All constraints are shown in Figure 7.

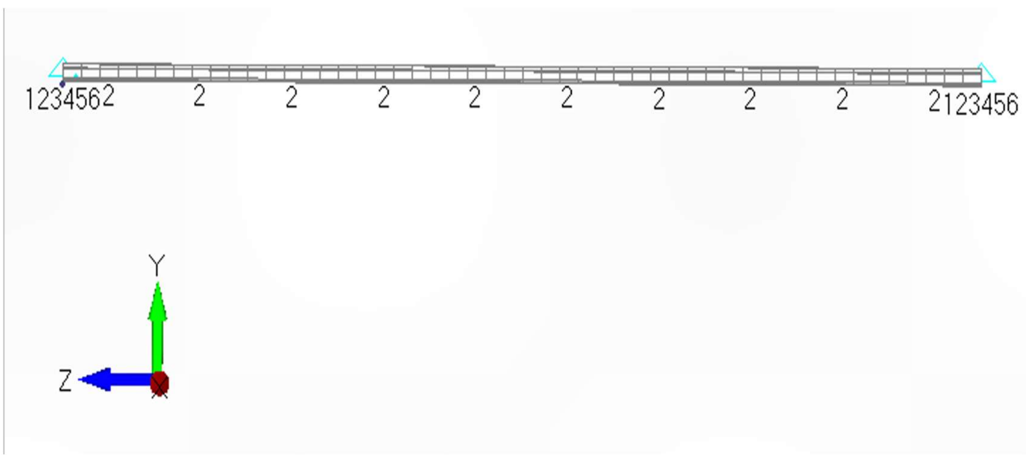


Figure 7. Rail constraints

After constraints and loads have been set, we can start the buckling analysis. From the analysis we get the eigenvalue which tells us how much more strain the rail can endure before buckling.

From the buckling analysis we get the eigenvalue for the rail which is 1.008056. this means that the compressive axial load that the rail is under is practically equal to its critical load. There for the rail can't take anymore compressive load before buckling. From the static analysis that is created automatically along with the buckling analysis we can see that the compressive axial force in the rail is -282124 N. From Figure 8 we can see an exaggerated deformation of the rail. When the rail starts to buckle it buckles around its profiles z-axis.

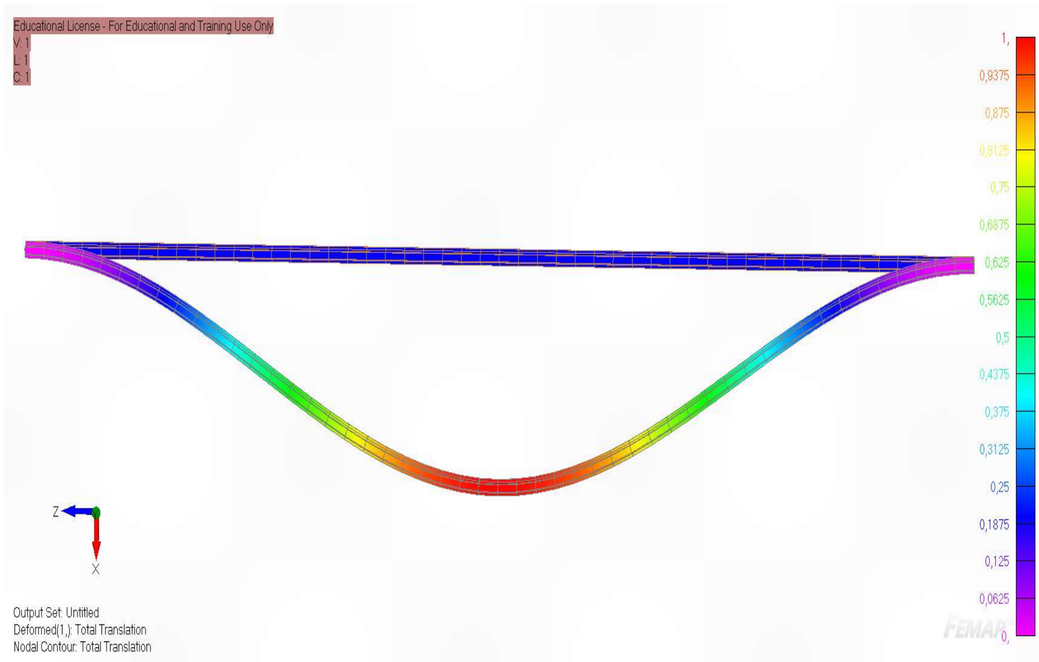


Figure 8. Deformed rail and undeformed rail

Table 4. buckling analysis values

Variable, unit	**Value**
Temperature change, °C	**15**
Compressive axial load affecting the rail, N	**-282 124**
Critical load factor n_{cr} (Eigenvalue)	**1,008056**
Critical load P_{cr}, N	**284 397**

In Table 4 we have gathered some of the important information gained from the buckling analysis. If the temperature would rise more and there would be no extra support to the rail or other measures to increase its stiffness, the rail would start to buckle. If we would put a support in the middle of the rail that would keep the center point fixed it would half the effective length and there for make the critical load four times as large than before.

4 Conclusions

As a solution method for buckling, FEMAP was found to be very comprehensive. It included translations, stresses, axial forces, eigenvalues etc. in its postprocessing data. FEMAPs results very visualized well for example color coded results in the 3D-model. MATLAB code was good for analyzing linear static case by giving the nodal displacements. The buckling analysis with MATLAB requires eigenvalue frequency analysis. From eigenvalue frequency analysis the eigenvalues and critical loads could be calculated. The problem with MATLAB code is describing the complicated cross-section such as the railway tracks cross-section profile. In MATLAB a simpler square column had to be used. There for FEMAP is more useful when solving buckling problems with complex column profiles. For this study MATLAB was extremely useful and time saving solution method when the problem was simple.

This study is useful guideline for a column designer to decide a solution method for a buckling problem. This study could be expanded to include for example the mass matrices, the eigenvalue frequency analysis and complex profiles in MATLAB. This way a very accurate comparison could be made between FEMAP and MATLAB as a buckling solution method.

References

Bathe, K. J. 2014. Finite Element Procedures. Second edition. pp. 90–92, 114, 630–636, 762, 839.

Bedford, A., Liechti, K. M. 2020. Mechanics of materials. Second edition.

Beer, F. P., Johnston, E. R., DeWolf, J. T., Mazurek, D. F. 2012. Mechanics of materials. Sixth edition. pp. 631–691.

British Steel, 2022. Profile selector - Vignole. [Internet materials]. [Cited 17.3.2022]. Available: https://britishsteel.co.uk/media/272091/60-e1.pdf.

FEMAP, 2021. Help: Basic structural analysis. [Computer software]. [Cited 14.4.2022]. Available: FEMAP software help section under Basic structural analysis.

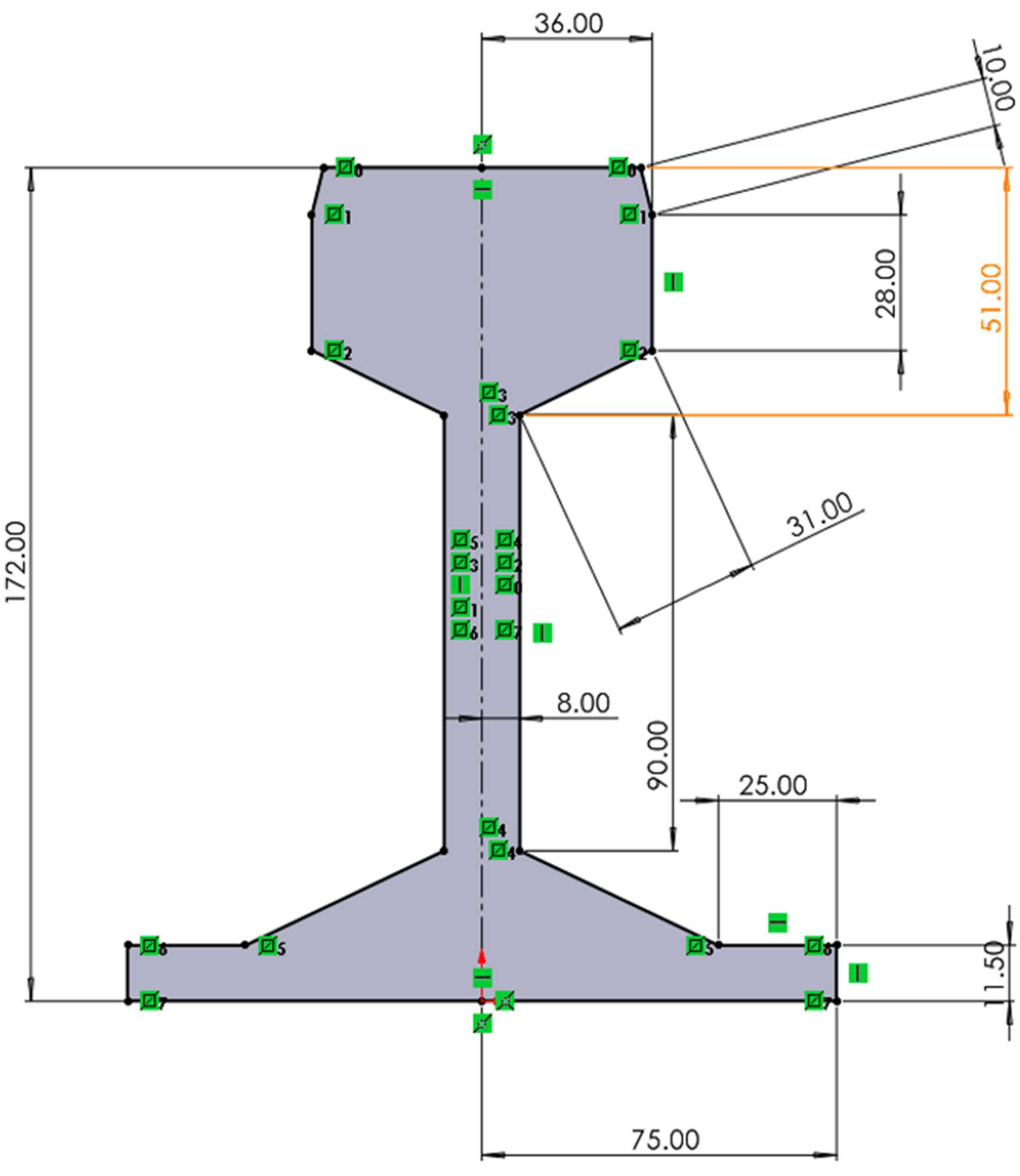
FEMCI Book, 1995. Applying preloads with the differential stiffness approach. [Internet materials]. [Cited 22.4.2022]. Available: https://femci.gsfc.nasa.gov/preloads/.

Kauppinen, M., 2011. Liikennevirasto, Ratakiskon elinkaari. [Internet materials]. [Cited 18.3.2022]. Available: https://julkaisut.vayla.fi/pdf3/lts_2011-/01_ratakiskon_elinkaari_web.pdf.

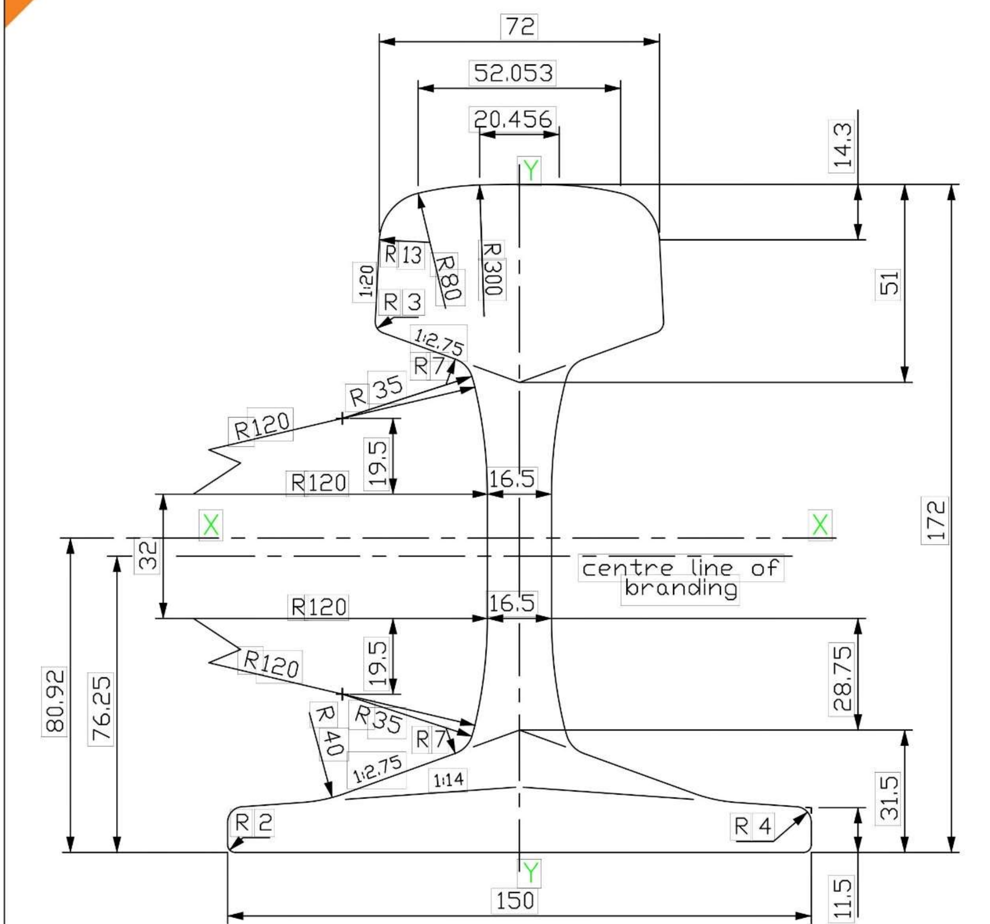
Kaya, A., Dawood, M., Gencturk, B. 2015. Repair of corroded and buckled short steel columns using concrete filled GFRP jackets. Construction and Building Materials. Volume 94. pp. 20–27.

Li, B., Cheng C., Song Z., Cao X. & Kong Z., 2022. Local buckling behavior of high strength steel welded box-section columns under axial compression. Thin-Walled Structures. Volume 171.

Mäkelä, M., Soininen L., Tuomola S., Öistämö J. 2020. Tekniikan kaavasto. 21. edition. Tampere, Tammertekniikka.



Appendix 1. Measurements of the rail profile



Rail: 60 E1 (UIC 60)	Standard	EN 13674-1	Moment of inertia Ixx (cm⁴)	3038.3
BRITISH STEEL	Area (cm²)	76.70	Moment of inertia Iyy (cm⁴)	512.3
	Density (g/cm³)	7.85	Section modulus base (cm³)	375.5
	Mass per metre (kg/m)	60.21	Section modulus head (cm³)	333.6
	Dimension A (mm)	20.456	Section modulus left of YY (cm³)	68.3
	Dimension B (mm)	52.053	Section modulus right of YY (cm³)	68.3

Appendix 2. UIC 60 rail profile (British steel 2022)