



# TRK LOYDU

## TECHNICAL CIRCULAR

**Circular No: S.P 01/19**

**Revision: 0**

**Page: 1**

**Adoption Date: Jan 2019**

**Related Requirement : TL/IACS CSR Pt.1, Ch.8, Sec. 5[2] .**

**Subject : Procedure for the Determination of the Ultimate Strength of Stiffened Panels by Using Non-Linear Finite Element Analysis**

**Entry into Force Date : 01.02.2019**

The following procedure is used where non-linear finite element analysis applied for determination of ultimate strength of stiffened panels. The procedure is developed in consideration to TL/IACS CSR Pt.1, Ch.8, Sec. 5 [2] particularly 2.2.4, however, basic principles set out in the procedure can also be applied where Non-linear Finite Element Analysis is required or used for determination of ultimate strength of stiffened panels either TL/IACS CSR (e.g. TL/IACS CSR Pt.1, Ch. 5, Sec.2, App. 2) or other Trk Loydu Rules.

# **TÜRK LOYDU**



**PROCEDURE FOR THE DETERMINATION OF THE  
STRUCTURAL CAPACITY OF STIFFENED PANELS BY USING  
NON-LINEAR FINITE ELEMENT ANALYSIS**

**FEBRUARY 2019**

**PROCEDURE FOR THE DETERMINATION OF THE STRUCTURAL CAPACITY OF STIFFENED  
PANELS BY USING NON-LINEAR FINITE ELEMENT ANALYSIS.**

1.	Introduction.....	1
2.	Terminology .....	2
3.	Theoretical background.....	3
4.	Implementation of NLFEA procedure.....	5
4.1	Model construction .....	6
4.2	Material .....	6
4.3	Boundary conditions.....	6
4.4	Load Application .....	8
4.5	Load shortening curve .....	8
4.6	Initial imperfections.....	10
4.7	Mesh .....	10
5.	Implementation of the procedure using ANSYS workbench.....	10
5.1	Solution of the initial static structural problem.....	11
5.2	Eigenvalue buckling .....	11
5.3	Preparation of the deformed model .....	12
5.4	Conversion of the deformed model in ANSYS workbench readable format .....	12
5.5	Solution of the non-linear problem and derivation of the LSC.....	12
5.6	Derivation of the LSC .....	13
6.	Non-Linear Controls .....	13
7.	Acceptance Criteria .....	14
7.1	Mesh fineness .....	14
7.2	Convergence acceptance criteria .....	14
7.3	Estimation of collapse point.....	15
8.	Required Qualifications of the User .....	15
9.	Worked examples.....	16
9.1.	Example No 1.....	16
9.2.	Example No 2.....	21
10.	References.....	25

## **PROCEDURE FOR THE DETERMINATION OF THE ULTIMATE STRENGTH OF STIFFENED PANELS BY USING NON-LINEAR FINITE ELEMENT ANALYSIS.**

### **1. Introduction**

The problem of the buckling capacity and the ultimate strength assessment of stiffened panels is addressed in CSR-H by using closed form analytical formulation. However, given the complexity of the problem and the several combinations that can occur in practice, CSR-H underline the necessity for the Classification Societies to have a clear and validated procedure for the assessment of the buckling strength assessment and the ultimate strength capacity of stiffened panels by using the Non-Linear Finite Element Analysis (NLFEA).

The term “Non-Linear Finite Element Analysis” covers a large number of analysis types for different purposes and objects. The content of this document is written with analyses of steel structures in mind and it is mainly referred to the assessment of stiffened panels subject to in-plane loads with/or without the simultaneous presence of lateral pressure. The aim of the present document is to provide guidance for the consistent application of NLFEA and particularly recommendations for:

- The preparation of the model
- The mesh generation
- The simulation of the material nonlinearity
- The applied boundary conditions
- The solution process
- The interpretation and presentation of the results

Since in TL the well established software package ANSYS is used for performing FEM calculations, the present guidance, although includes generic topics which are applied independently of the used software, in particular paragraphs special emphasis is given to relevant ANSYS features for NLFEA.

NLFEA is today the state of the art recognized methodology to predict the structural ultimate/buckling behaviors of stiffened panels more precisely. However, since NLFEA is a quite complex method and very sensitive on the calculation of the influencing parameters, such as boundary conditions, assumption of initial imperfection, etc. it is evident that, as far as possible, specific guidelines are needed for the consistent application of the NLFEA.

Extensive effort in this document was also given to the validation of the procedure through benchmarking with published numerical calculations and/or experimental results.

## 2. Terminology

- **Stiffeners** are called the smaller reinforcing sections of the stiffened panel.
- **Frames** are called the larger stiffening members upon which the smaller are supported. Their direction is perpendicular to that of stiffeners.
- **Plate** the metallic sheet upon which stiffeners and frames are welded.
- **Stiffened panel** is the assembly of plate with the associated stiffeners and frames.
- **Load Shortening Curve** of a panel is a diagram indicating the relationship between the applied compressive load (y-axis) and the corresponding resulting shortening (x-axis) in the same direction.
- **Normalized Shortening Curve** of a panel is the Load Shortening Curve where the applied Load is expressed as the applied average compressive stress  $\sigma_{ave}$  divided by the yield stress of the material  $\sigma_y$  and the shortening is expressed as the ratio of the average strain ( $\epsilon_{ave}$ ) divided by the strain at yield point ( $\epsilon_y$ ).
- **Global coordinate system** is a right handed cartesian coordinate system the origin of which coincides with one of the four vertices of the plate having the x-axis parallel with the stiffeners, the y-axis parallel to the frames and the z-axis perpendicular to the plate surface.

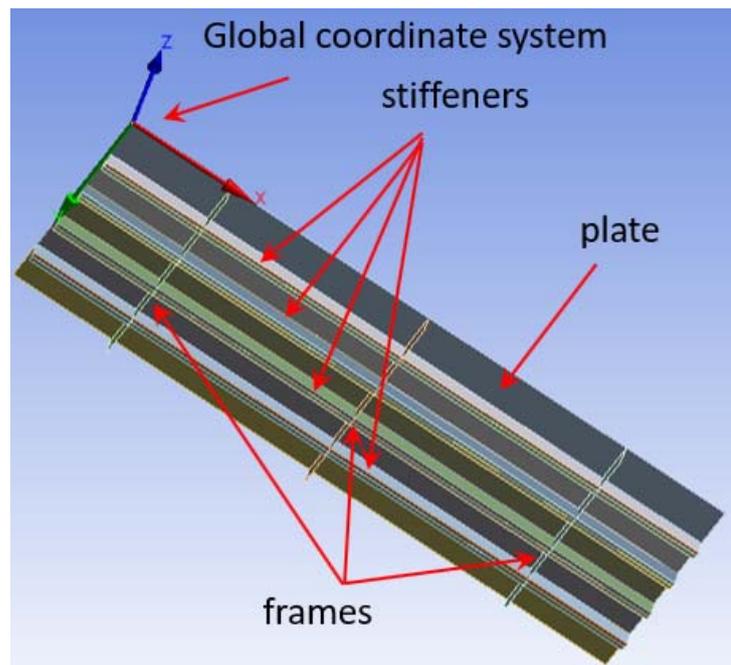


Figure 2.1: Terminology

### 3. Theoretical background

To simulate buckling/plastic collapse behavior of stiffened plates, both analytical and numerical methods can be applied. With the numerical methods, such as finite elements method (FEM), it is possible to simulate structural behaviors considering both material and geometrical nonlinearities. Material non-linearities may stem from the non-linear stress-strain relationship of metallic materials mainly in the plastic region. Geometrical non-linearities are related to the large deflections noted during the post-buckling phase and the strong influence of non-linear terms in the relevant governing equations.

Modern software packages, like ANSYS, can address both types of non-linearities. Many different forms of material nonlinearity can be represented in ANSYS, including hyperelasticity, rate and temperature dependant plasticity, pressure dependent plasticity, material strength degradation (damage), etc. As regards the material non-linearity in the NLFEA for stiffened panels, it is usually addressed through the adoption of an appropriate bi- or tri-linear stress-strain model curve which approximates the actual behavior of the material in the plastic region. Of course, more complex material behavior, other than the piecewise linear model, could be also adopted, but against the computational cost. Geometrical nonlinearities are addressed through an iterative convergent solution scheme, in which external loads are gradually applied in a stepwise iterative process. The number of the iterative steps can be defined by the user or automatically be determined by the software.

The solution to the nonlinear governing equations can be achieved through an incremental iterative approach. The incremental form of the governing equations can be written as:

$$K(u) \cdot \Delta u = \Delta P$$

in which  $\Delta u$  and  $\Delta P$  represent the unknown incremental displacement vector and the known incremental applied load vector, respectively. The solution is constructed by performing a series of linear steps in the appropriate direction in order to closely approximate the exact solution. Depending on the nature and degree of the nonlinearity, the magnitude of each step and its direction may involve several iterations. The computational algorithms and the associated parameters must be chosen with extreme care. The solution to non-linear problems may not be unique.

The Newton-Raphson (N-R) iterative scheme has been adopted in ANSYS software package. In this approach, the iterations start with a trial (assumed) solution:

$$u = u_i,$$

to determine the magnitude of the next step (increment):

$$\Delta u_i = K^{-1}(u_i) \cdot \Delta P,$$

and the corresponding out-of-balance load vector:

$$\Delta R_i = \Delta P - K(u_i) \cdot \Delta u_i,$$

which is the difference between the applied loads and the loads evaluated based on the assumed solution.

In order to satisfy the equilibrium conditions exactly, the out-of-balance load vector  $\Delta R_i$  must be zero. However, as the nonlinear equilibrium conditions are solved approximately, a tolerance is introduced for the out-of-balance load vector in order to terminate the solution procedure. In each iteration, the Newton-Raphson method computes the out-of-balance load vector and checks for convergence based on the specified tolerance. If the convergence criterion is not satisfied, the trial solution is updated as:

$$u_{i+1} = u_i + \Delta u_i$$

based on the calculated incremental displacements, and the next incremental solution vector is determined as:

$$\Delta u_{i+1} = K^{-1}(u_{i+1}) \cdot \Delta P$$

leading to the computation of the new out-of-balance load vector:

$$\Delta R_{i+1} = \Delta P - K(u_{i+1}) \cdot \Delta u_{i+1};$$

this procedure is repeated until convergence is accomplished.

Several methods for improving the convergence (or convergence rate) are available in ANSYS. These include automatic time stepping, a bisection method, and line search algorithms. The user may choose to have full control or let ANSYS choose the options.

In a nonlinear solution in ANSYS the iterative scheme is structured in three distinct levels which are defined as:

- (1) Load Steps,
- (2) Substeps, and
- (3) Equilibrium Iterations.

The number of load steps is either specified by the user or automatically determined by the software. Different load steps must be used if the loading on the structure changes abruptly. The use of load steps also becomes necessary if the response of the structure at specific points in time is desired. A solution within each load step is obtained by applying the load incrementally in substeps. Within each substep, several equilibrium iterations are performed until convergence is accomplished, after which ANSYS proceeds to the next substep. As the number of substeps used increases, the accuracy of the solution improves. However, this also means that more computational time is being used. ANSYS offers the **Automatic Time Stepping** feature to optimize the task of obtaining a solution with acceptable accuracy in a reasonable amount of time. The automatic time stepping feature decides on the number and size of substeps within load steps. When using automatic time stepping, if a solution fails to converge within a substep, the bisection method is activated, which restarts the solution from the last converged substep.

Nonlinear analyses require more computational time. Therefore, when solving nonlinear static problems, it is recommended to solve a preliminary version of the problem with no nonlinearities before proceeding to the non-linear problem. The results from the linear solution may reveal mistakes in modeling, meshing, and application of boundary conditions in a shorter time frame. Also, the linear solution provides information about the regions where high stress gradients are expected, thus guiding the user to modify the mesh (make it more refined) in those regions.

In nonlinear analyses, it is important to utilize all possible simplifications in order to improve convergence and reduce the computational cost. For example, if the problem can be simplified as a plane stress or plane strain idealization, then the user should take advantage of this opportunity.

#### 4. Implementation of NLFEA procedure

NLFEA Analyses are performed in two steps. First, a linear Eigen-value analysis is carried out in order to create data for the initial imperfection which later will be used to trigger the nonlinear buckling of the stiffened plate. The eigen-modes calculated from the linear buckling analysis are used for the preparation of the “initially deformed” model which will be used as input in the non-linear analysis. The target of the NLFEA is mainly to determine the ultimate capacity of the panel which can be defined as the collapse load, i.e. the maximum load that the panel can withstand.

The main steps of the Analysis are shown in Figure 4.1.

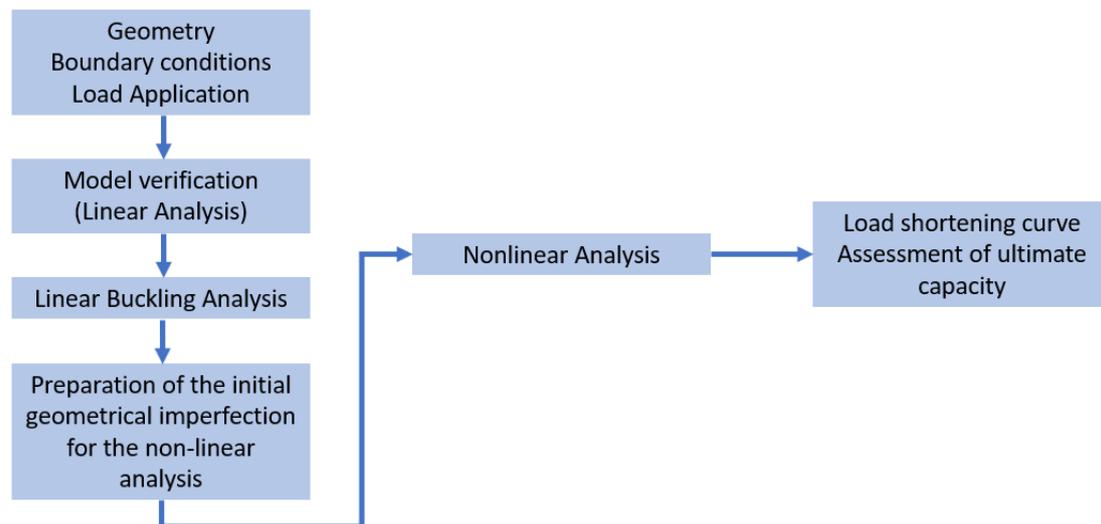


Figure 4.1: The applied approach

## 4.1. Model construction

### 4.1.a. Element types

Plates, stiffener webs and stiffener flanges, as well as frame webs and flanges are modelled by using **4-node shell elements**. Higher order elements, in general, are not necessary to be used, since the obtained accuracy by using 4-node shell elements is considered satisfactory.

### 4.1.b. Model extent

For prevention of initiation of collapse of the panel at the model boundaries, in case of cross-stiffened panels, the model is recommended to represent three frame spans ( $1/2+1+1+1/2$ ) along the direction of the stiffeners. In the transverse direction the model is recommended to include 5 stiffeners, i.e. a total breadth of six stiffener spans. The transverse frames, if their rigidity is large, could be reflected by implementing appropriate boundary conditions at the nodes which coincide with the trace of the frames on the plate surface instead of including them in the model.

## 4.2. Material

In case of absence of relevant experimental data, the stress-strain curve of the material, in case of mild steel, shall be approached by a bi-linear material model, including strain hardening through the adoption of a tangent modulus equal to 1000 N/mm<sup>2</sup>. Young's modulus and Poisson ratio shall be taken as 206,000 N/mm<sup>2</sup> and 0.3, respectively.

If experimental data exist a more accurate approximation of the stress-strain curve can be adopted.

In case of material other than steel, the stress-strain curve shall be approached by an appropriate piecewise linear behavior.

## 4.3. Boundary conditions

Appropriate boundary conditions of displacement type (restrained displacements -free-rotations) are to be applied to the edges of the stiffened panel. As a general Rule the out-of-plane displacement of the edges of the stiffened panel will be restrained to all edges of the plate.

The nodes of the end cross sections of the stiffeners/frames are subject to the same boundary conditions as the corresponding plate edges.

General guidance for the implementation of the appropriate boundary conditions for each load case is given in Table 4.1. In case of convergence difficulties, the recommended boundary conditions may be differentiated.

In case that the frames (primary supporting members) are of excessive rigidity compared to stiffeners, the structural modelling of the frames can be avoided by replacing them with the restriction of the displacement in the z-direction of the plate nodes located along the trace of the frame on the plate surface.

**Table 4.1: Boundary conditions.**

<p>Compression in x-direction</p>	
<p>Compression in y-direction</p>	
<p>Simultaneous compression in x and y directions</p>	
<p>Shear loading along the x-direction</p>	

#### 4.4. Load Application

For the derivation of the load shortening curves and the estimation of the buckling capacity the forcing is introduced in the form of a forced displacement of adequate size of the edge nodes of the model and the corresponding compressive or shear force is calculated as the corresponding reaction force in the opposite direction. The size of the imposed forced displacement shall be adequately large in order to produce deformations beyond the ultimate buckling capacity of the panel.

Similar procedure is followed also for the cases of transverse in-plane compressive loads and tangential in-plane shear loads.



Figure 4.2: Method of load application

#### 4.5. Load shortening curve

The Load Shortening Curve (LSC) is a diagram representing the unidirectional shortening (as abscissa) of a stiffened or unstiffened panel due to the application of a compressive load in the same direction (see Figure 4.3) (as ordinate). The LSC can be presented in several alternative formats, either dimensional (e.g. Applied Load in Newtons vs Displacement in mm) or non-dimensional (normalized) eg. (average compressive stress  $\sigma_{ave}$ )/(yield stress  $\sigma_y$ ) vs (average strain  $\epsilon_{ave}$ )/(strain at yield  $\epsilon_y$ ). In the present report the LSC is represented in the nondimensional format (see Figure 4.4), where:

- average compressive stress  $\sigma_{ave} = F / A$
- average strain  $\epsilon_{ave} = dL/L$ ,
- F is the applied force or the equivalent corresponding reaction force
- A is the cross sectional area (including plate and stiffeners) upon which the compressive force is applied,
- dL is the shortening of the plate.

The peak (maximum) of the LSC indicates the start of the collapse of the panel since steep degradation of the resistance of the panel to the applied compressive stress follows just after. Same point defines the ultimate strength or ultimate buckling capacity of the stiffened panel.

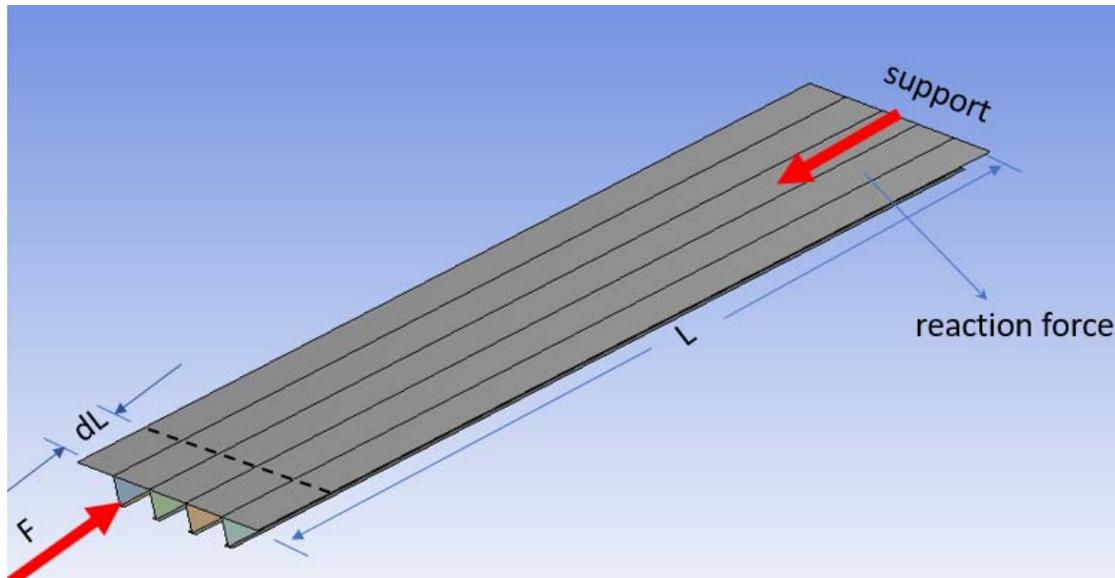


Figure 4.3: Load shortening terminology

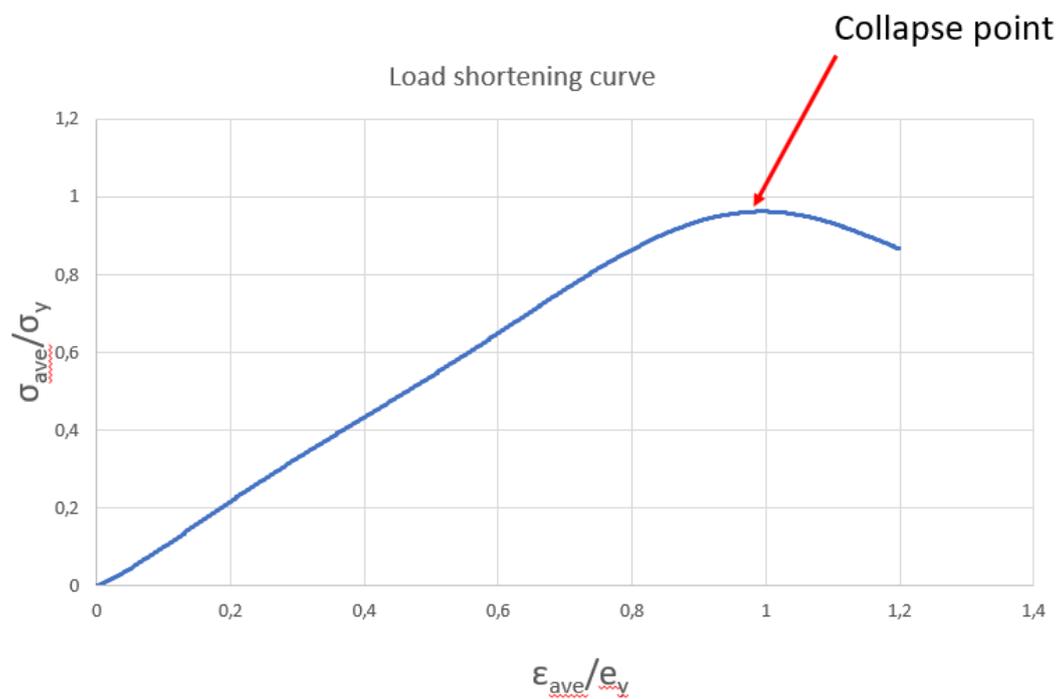


Figure 4.4: Normalized Load Shortening Curve

#### **4.6. INITIAL IMPERFECTIONS**

A geometrical imperfection is used to trigger the non-linear response of the panel and is normally required to enable the estimation the buckling strength of the panel by using the nonlinear finite element method. In most of the cases the first eigenvalue buckling mode can be used as initial imperfection. In case that the first eigen-mode cannot trigger the non-linear buckling, a higher mode appropriately scaled, could be used as initial imperfection.

#### **4.7. MESH**

In Finite Element Method, the software discretizes a structural continuum into finite elements. Governing differential equations are then solved numerically for each element. Different strategies are implemented to account for both geometrical and physical non-linearities. In the academic context, a shell is a structure whose structural behaviour can be mathematically described by shell elements. A generalized shell element is a 2-dimensional element that can withstand both, in-plane and out-of-plane forces and moments.

The mesh has to be modeled by using 4 node shell elements. The size of the mesh shall be adequately fine in order to reflect the localized deformations and stress distributions developed during buckling. The mesh size in general depends on the geometric configuration of the stiffened panel and the applied loading. The following rules shall be followed for the determination of the appropriate mesh size:

- The aspect ratio of the elements shall be as close to unity as possible.
- The minimum number of elements between the stiffeners shall be eight.
- Minimum three elements shall be used across the web depth of the stiffener.
- Minimum two elements shall be used across the flange width of the stiffeners, in case of T stiffeners and one element in case of angles and bulb profiles.

### **5. Implementation of the procedure using ANSYS workbench**

The well-established code ANSYS Workbench is to be used for the production of the LSC and the estimation of the ultimate buckling capacity of stiffened panels by using Non-Linear Finite Elements. The whole procedure consists of 5 steps as described below (see the typical Project Schematic view in Figure 5.1):

1. Solution of the initial static structural problem.
2. Solution of the eigenvalue buckling problem
3. Transfer of the buckling analysis results to the ANSYS Mechanical APDL for the preparation of the deformed model, by using the UPGEOM command.
4. Conversion of the deformed model to ANSYS workbench readable format (e.g. Parasolid).
5. Solution of the non-linear problem and derivation of the LSC.

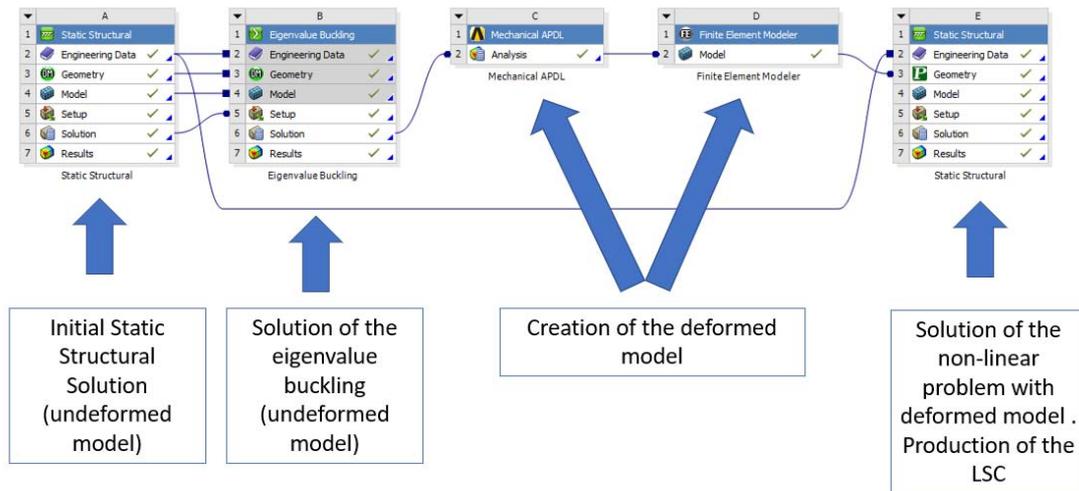


Figure 5.1: The ANSYS Workbench Project Schematic

## 5.1 Solution of the initial static structural problem

This step is implemented for two purposes:

- In order to check the model (e.g. elements connectivity) and the applied boundary conditions (e.g. elimination of any rigid body motions), and,
- In order to prepare the necessary input for the solution of the eigenvalue buckling problem.

At this stage the model has to be constructed to the necessary detail and the mesh to have the same density as the one that will be used in the final solution by using non-linear analysis. The applied boundary conditions and the mechanical properties of the material shall be also kept the same with those used later in the NL solution. In this step, atypical load(s) has(-ve) to be applied (in the form of forced displacements as previously explained), in the same way as in the final stage. The geometrical nonlinearities need not to be addressed at this stage (The “**Large Deflection**” solution parameter can be turned to “**Off**”).

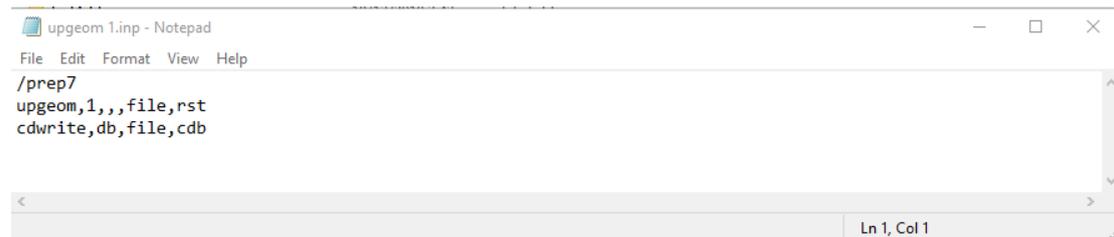
## 5.2 Eigenvalue buckling

Based on the solution of the static structural problem described in Step (1), the eigenvalue buckling problem is solved and the first eigenvalue modes are produced in order to be used as the initial imperfection which will trigger the non-linear buckling at the final step. In general, the first eigenmode shall be adopted as initial imperfection. However, if due to the applied boundary conditions and/or the geometrical configuration, the first eigenmode is of a very local form, the first eigenmode which produce a global deformation should be selected instead.

### 5.3 Preparation of the deformed model

In order to prepare the deformed (imperfect) model which will be used in the non-linear analysis the solution of Step No 2 is transferred to the ANSYS Mechanical APDL software where the command UPGEOM is inserted as input file.

A typical input file (\*.inp) for the execution of the UPGEOM command is shown in Figure 5.2.



```
upgeom 1.inp - Notepad
File Edit Format View Help
/prep7
upgeom,1,,,file,rst
cdwrite,db,file,cdb
Ln 1, Col 1
```

Figure 5.2: Syntax of a typical input file (\*.inp) with scaling factor 1 for the implementation of the UPGEOM command

### 5.4 Conversion of the deformed model in ANSYS workbench readable format

In order the deformed model to be converted to a format readable by ANSYS Workbench the deformed model produced in step No 3 is transferred to the FE Modeler routine, where via the Command “**Update**” the deformed geometry is converted to a format which is readable by the Workbench Design Modeler or Spaceclaim.

### 5.5 Solution of the non-linear problem and derivation of the LSC

After the completion of Step No 4, the deformed model in Parasolid Format is introduced to a new Static Structural module. The Material properties (through the Engineering Data Module) remain as in Step 1. Due to the deformed shape of the model the connectivity of the model has to be checked and appropriate corrections to be made. Mesh of the same size as in the run of Step 1 is created for the deformed shape. The boundary conditions are applied in the same way as in Step No 1.

The forcing is inserted as a forced nodal displacement of all nodes located on the corresponding panel edge (including end nodes of stiffeners) and the applied force is calculated through the assessment of the corresponding reaction force in the same direction as the forced displacement. The applied forced displacement shall be of adequate magnitude so that the collapse of the panel to be reached. Successively increased forced displacements may need to be applied until the collapse point will be obtained.

In Step No 5, the **Large Deflection** Parameter has now to be turned to “**On**”.

Following results shall be retrieved from the solution output:

- The total deformation of the panel
- The directional deformation of the loaded edge in the direction of the forced displacement.
- The reaction force developed by the model supports, in the same direction of the applied force.
- The Equivalent Stress (in order to check extend of yielding).

## 5.6 Derivation of the LSC

Upon the convergence of the solution process, the normalized LSC is derived and plotted as described in Section 4.5.

## 6. Non-Linear Controls

As mentioned in Section 3, the non-linear solution of static structural problems is obtained through the implementation process based on the Newton-Raphson iterative scheme. The rate of convergence of this iterative process depends on the degree of the non-linearity that each examined problem presents. The improvement of the convergence may be obtained by the adjustment of several numerical parameters, the optimum value of which depends on the nature of the specific problem and the evolution of the convergence process.

ANSYS software permits the user to intervene in the solution process by adjusting a number of numerical parameters. Of course, the software can also handle these numerical parameters determining each time their optimum values. Since the adjustment of the convergence parameters shall be made by an experienced user, it is strongly recommended, the control of the convergence process to be left initially to the software by setting the “**Program controlled**” option for the several parameters.

In case that convergence difficulties are encountered, the following recommendations could be followed in order to improve the convergence:

1. The **Newton-Raphson Option** under the **Analysis Settings/Non-linear Controls** shall be set to the “**Unsymmetric**” value. Sometimes this adjustment could improve the convergence.
2. For highly nonlinear problems, the parameters “**Displacement Convergence**” and “**Rotation Convergence**” under the “**Analysis Settings/Non-linear Controls**” may be activated.
3. The parameter “**Line Search**” under the “**Analysis Settings/Non-linear Controls**” group of parameters, if activated, may enhance the convergence but very often it requires additional computational time.

Convergence difficulty due to an unstable problem is usually the result of a large displacement for small load increments. Nonlinear stabilization technique can help achieve convergence, particularly in the post buckling region. Nonlinear stabilization can be thought of as adding artificial dampers to all of the nodes in the system. Any degree of freedom that tends to be unstable has a large displacement causing a large damping/stabilization force. This force reduces displacements at the degree of freedom so stabilization can be achieved.

There are three options for controlling nonlinear stabilization:

- **Off** - Deactivates stabilization (default setting).
- **Constant** - Activates stabilization. The energy dissipation ratio or damping factor remains constant during the load step.
- **Reduce** - Activate stabilization. The energy dissipation ratio or damping factor is reduced linearly to zero at the end of the load step from the specified or calculated value.

There are two options for the **Method** property for stabilization control:

- **Energy** - Use the energy dissipation ratio as the control parameter (default setting).
- **Damping** - Use the damping factor as the control parameter.

When **Energy** is specified, an **Energy Dissipation Ratio** needs to be entered. The energy dissipation ratio is the ratio of work done by stabilization forces to eliminate potential energy. This value is usually a number between 0 and 1. The default value is 1.0e-4.

When **Damping** is specified, a **Damping Factor** value needs to be entered. The damping factor is the value that the ANSYS solver uses to calculate stabilization forces for all subsequent substeps. This value should be greater than 0.

## 7. Acceptance criteria

### 7.1 Mesh fineness

The mesh fineness has to comply with the requirements stated in Section 4.7.

### 7.2 Convergence acceptance criteria

As explained, in finite element analyses, problems involving nonlinearity are solved through iterations. These nonlinearities arise through the material behavior and the large deflections. The “correct” solution is approached in small *steps* in the application of the loads, referred to as convergence iterations. At the end of each iteration, ANSYS checks whether the solution satisfies a “built-in” convergence criterion. If the criterion is not satisfied, the last step is repeated with a smaller step size. This is repeated until the convergence criterion is satisfied. However, there are limits on the number of convergence iterations and, if a converged solution is not achieved within those limits, ANSYS terminates the solution process.

The following values are recommended to be used as convergence tolerances:

- Force Convergence: 0,5%
- Moment Convergence: 0,5%
- Displacement Convergence: 0,5%
- Rotation Convergence: 0,5%

When stabilization is used in order to facilitate convergence the maximum Energy Dissipation ratio shall be set to 0,0001.

### **7.3 Estimation of collapse point**

In order to consider that the collapse point has been reached and the post-buckling region has been entered, the slope of the Load Shortening Curve shall remain equal to or less than zero for a minimum range equal to at least 10% of the strain at yield  $\epsilon_y$ .

## **8. Required Qualifications of the User**

The person who is going to implement this procedure shall fulfill the below qualifications:

- The user shall possess the necessary theoretical background in the fields of Structural and Numerical Analysis in order to understand in depth the implementation of the present procedure.
- The user should be familiar with FE methodology, in general, and especially with the application of non-linear FE approach.
- The user has to be an experienced user of the ANSYS Workbench software package.

## 9. Worked examples

For the better understanding and embedding of the current procedure and its benchmarking, two worked examples are presented in the present Section.

### 9.1. Example No 1

The model used in this Example has been also assessed in the following LR design procedure:

*“Non-linear Structural Collapse Analysis for Plates and Stiffened Panels”, ShipRight Design and Construction- Additional Design Procedures, May 2016.*

#### a) Geometrical configuration

An orthogonal steel plate having the following dimensions:

- Length  $a$ : 12900 mm
- Breadth  $b$ : 4075 mm
- Thickness  $t_p$ : 17,8 mm

is stiffened in the longitudinal dimension with four T-shaped, equally spaced stiffeners, in the direction parallel to its longest side. The size of the stiffeners is:

- Web:  $h_w \times t_w = 463 \text{ mm} \times 8 \text{ mm}$
- Flange:  $h_f \times t_f = 172 \text{ mm} \times 17 \text{ mm}$
- Stiffener spacing  $s = 815 \text{ mm}$

The plate is also transversely stiffened with three rigid frames spaced 4300mm.

The geometrical configuration is depicted in Figure 9.1.

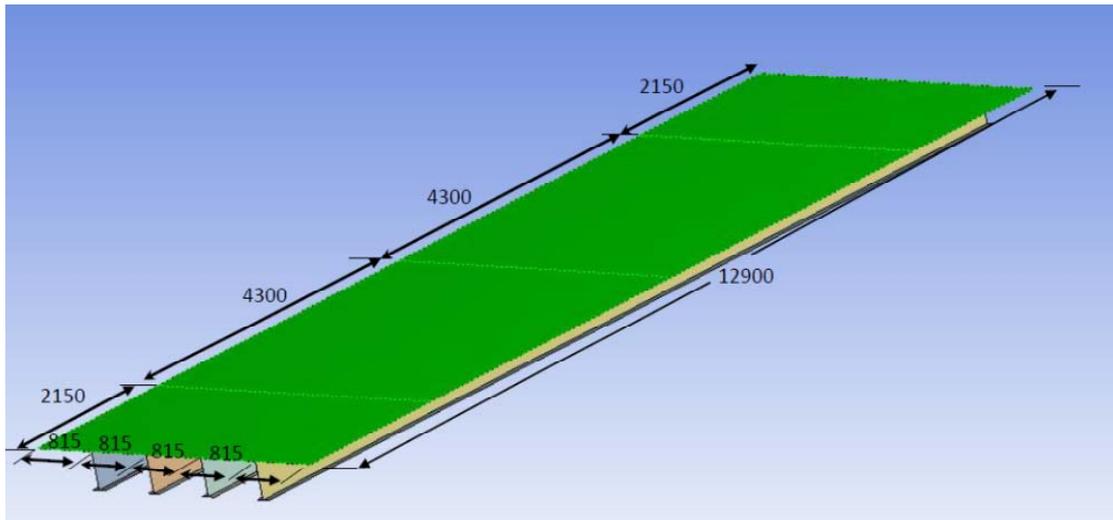


Figure 9.1: The geometrical configuration

### b) Material Properties

Both, plate and stiffeners are made of structural steel, with the following bilinear elastic characteristics:

- Yield stress:  $\sigma_y=315$  MPa
- Strain at yield  $\epsilon_y=0.001531$
- Young's Modulus:  $E=205.8$  GPa
- Tangent Modulus:  $E_t= 1000$  MPa

The stress-strain curve of the used steel is depicted in Figure 9.2.

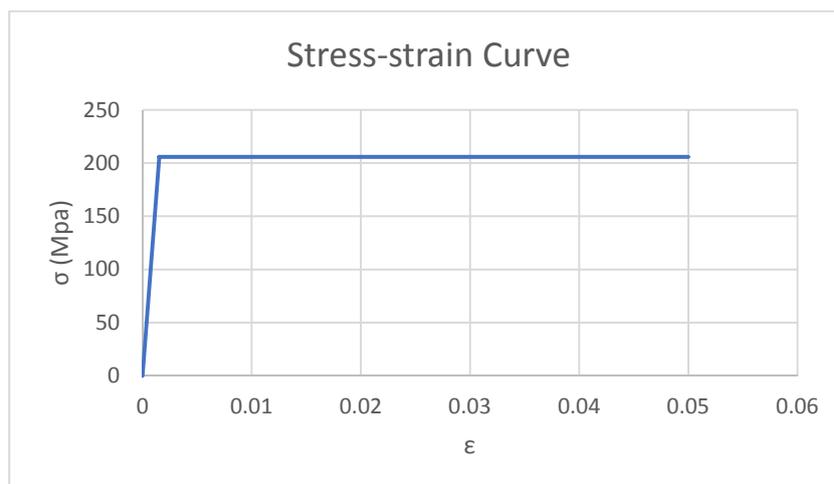


Figure 9.2: The stress-strain curve

### c) Solution process

The solution process is shown in Figure 9.3:

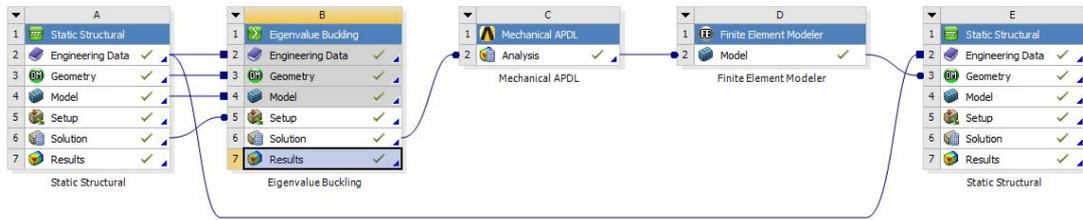


Figure 9.3: The project schematic

In order to prepare the deformed shape which will be used as initial imperfection, the linear buckling solution is obtained. The first linear eigenvalue buckling mode is shown (in scale) in Fig. 9.4a and 9.4b.

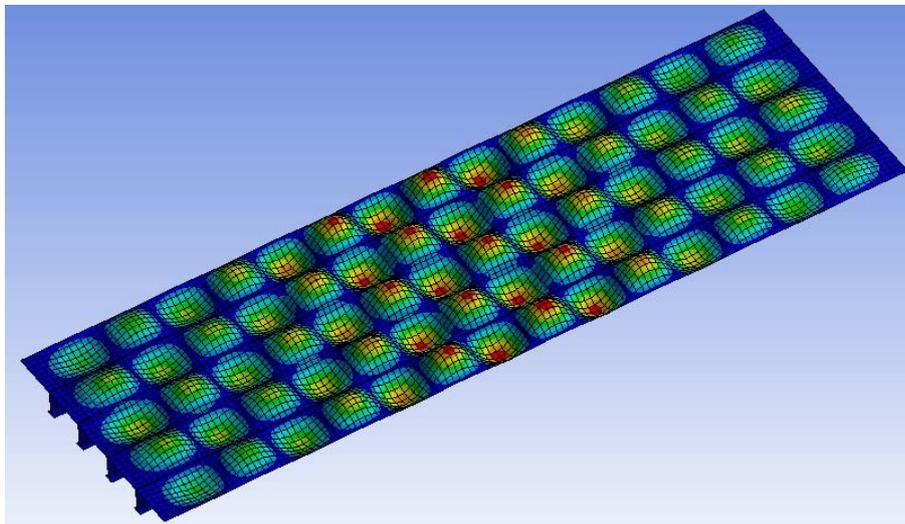


Figure 9.4a: The first linear eigenvalue buckling mode – Plate view

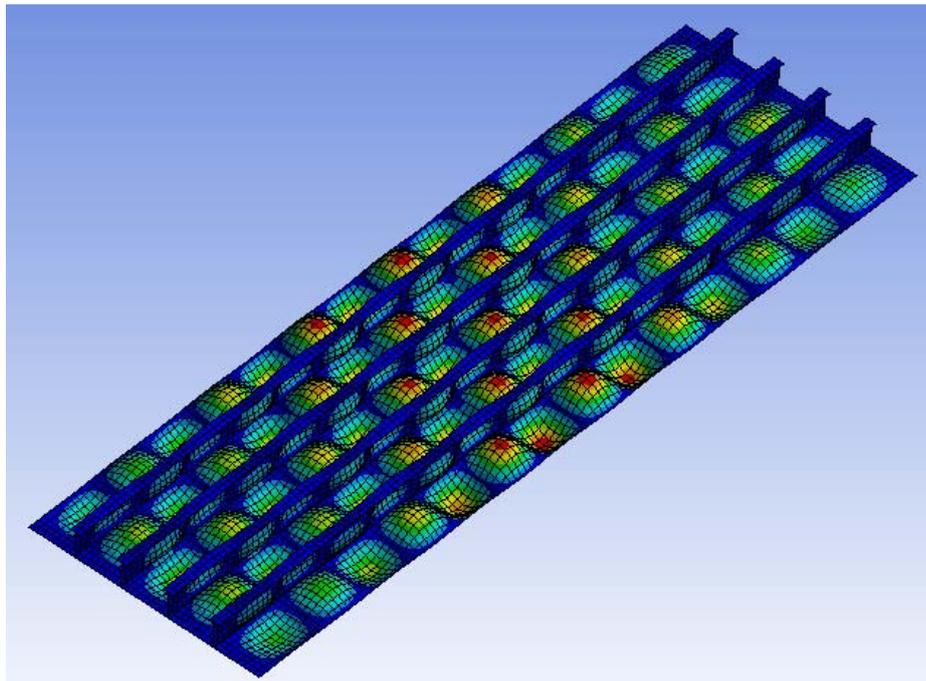


Figure 9.4b: The first linear eigenvalue buckling mode – Stiffener view

Next, the UPGEOM command of the ANSYS MECHANICAL APDL is used in order to prepare the “deformed” model according to the first eigenvalue mode, as described in Section 5. The model is then introduced to a new Static Structural Module and the non-linear solution is obtained through the iterative process described in Section 3.

In Figures 9.5a to 9.5c the buckled structure, after the non-linear solution is indicated.

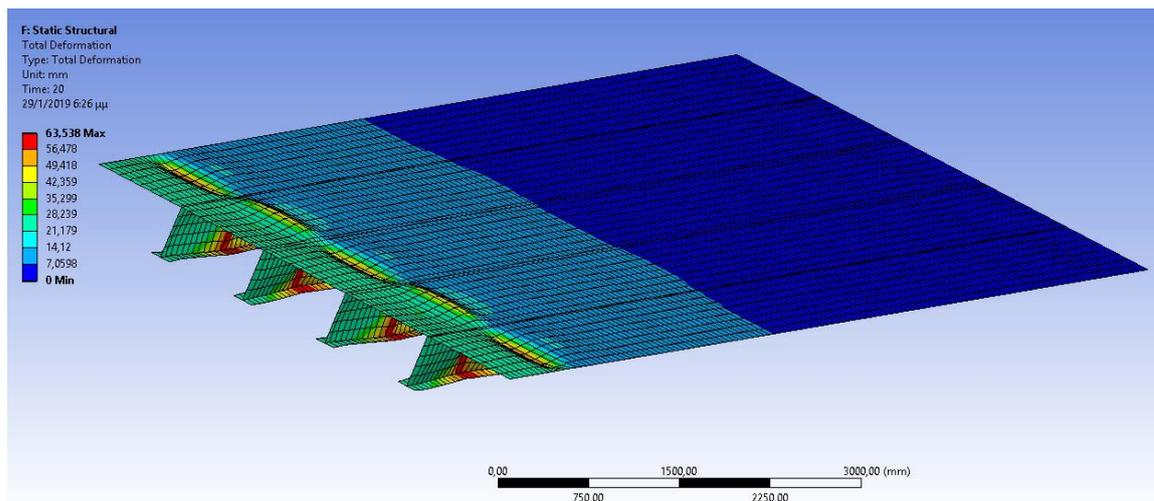


Figure 9.5a: The buckled structure – Plate view

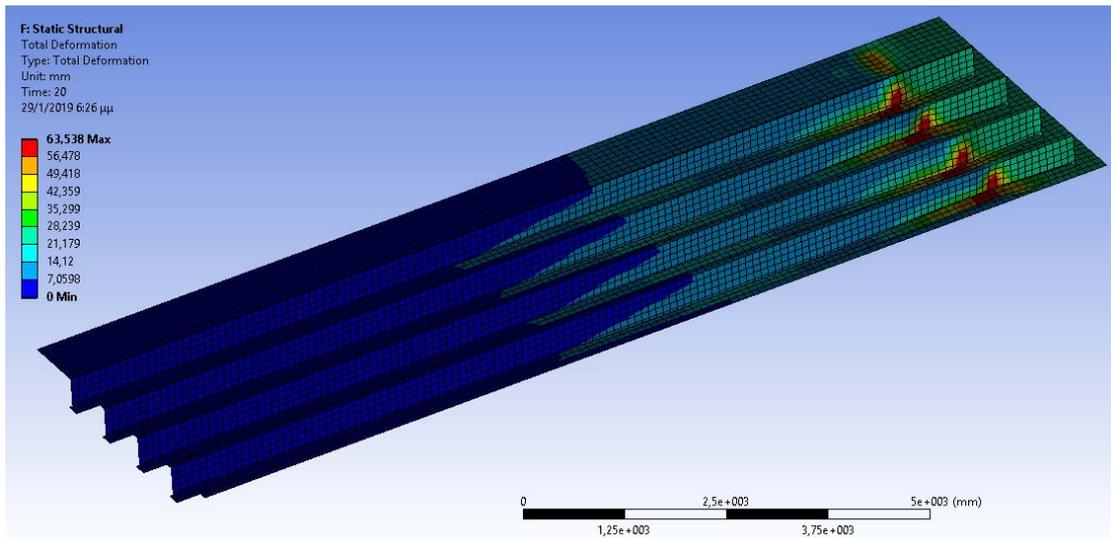


Figure 9.5b: The buckled structure – Stiffener view

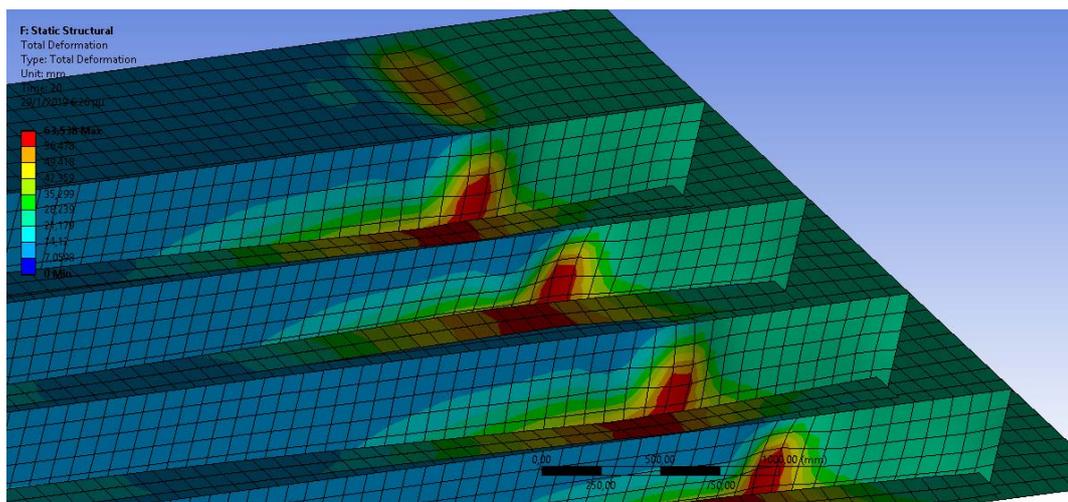


Figure 9.5c: The buckled structure – Stiffener view (detail)

In Figure 9.6 the resulting normalized Load Displacement curve is shown. From the shape of the curve it follows that the collapse of the panel occurs when the average normal compressive stress along the short edge of the stiffened plate approaches the value:

$$\sigma_{\text{collapse}} = 0,909 \times \sigma_y = 286 \text{MPa}.$$

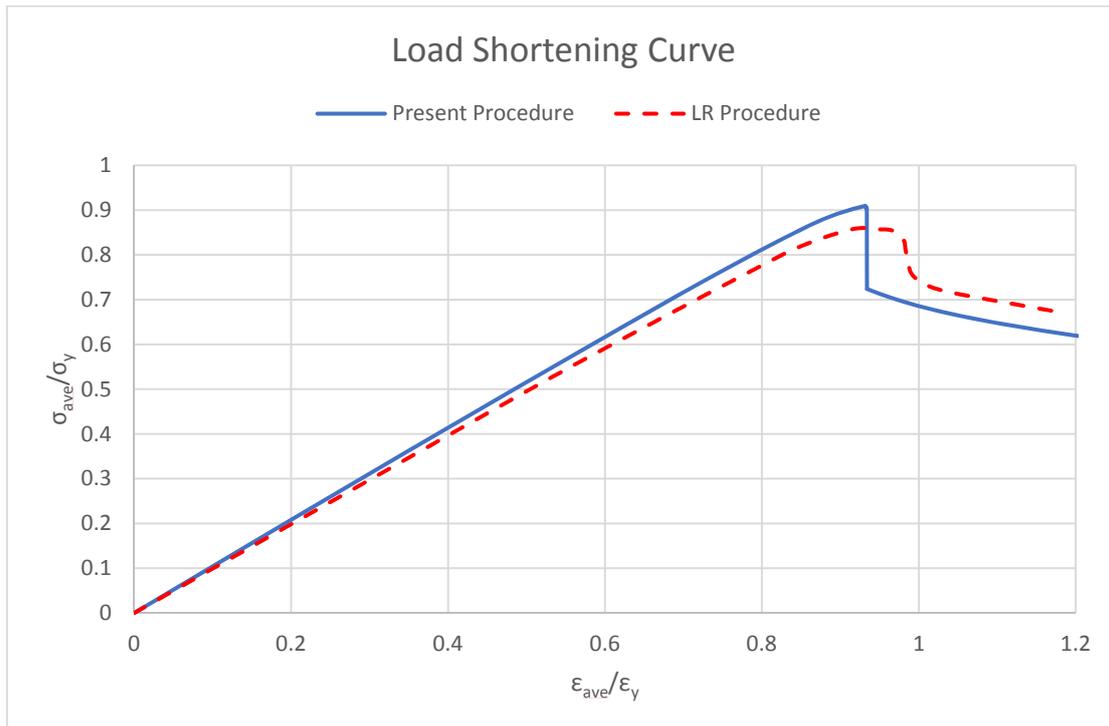


Figure 9.6: The normalized Load Shortening Curve

In Figure 9.6 a comparison is also made with corresponding results taken by the referenced LR procedure. Taking into account the fact that different software has been used in the two cases the comparison is considered satisfactory.

## 9.2 Example No 2

In order to verify the proposed procedure a comparison is attempted with relevant published results as included in the following publication:

*Imtaz Khan, Shengming Zhang, "Effects of welding-induced residual stress on ultimate strength of plates and stiffened panels", Ships and Offshore Structures, Vol. 6, No. 4, pp 297-309, 2011.*

### a) Geometrical configuration

An orthogonal steel plate with the following dimensions (Figure 9.7):

- Length a: 4300 mm
- Breadth b: 4075 mm
- Thickness  $t_p$ : 17,8 mm

is stiffened with four T-shaped, equally spaced stiffeners, in the direction parallel to its longest side. The size of the stiffeners is:

- Web:  $h_w \times t_w = 463 \text{ mm} \times 8 \text{ mm}$

- Flange:  $h_f \times t_f = 172 \text{ mm} \times 17 \text{ mm}$
- Stiffener spacing  $s = 815 \text{ mm}$

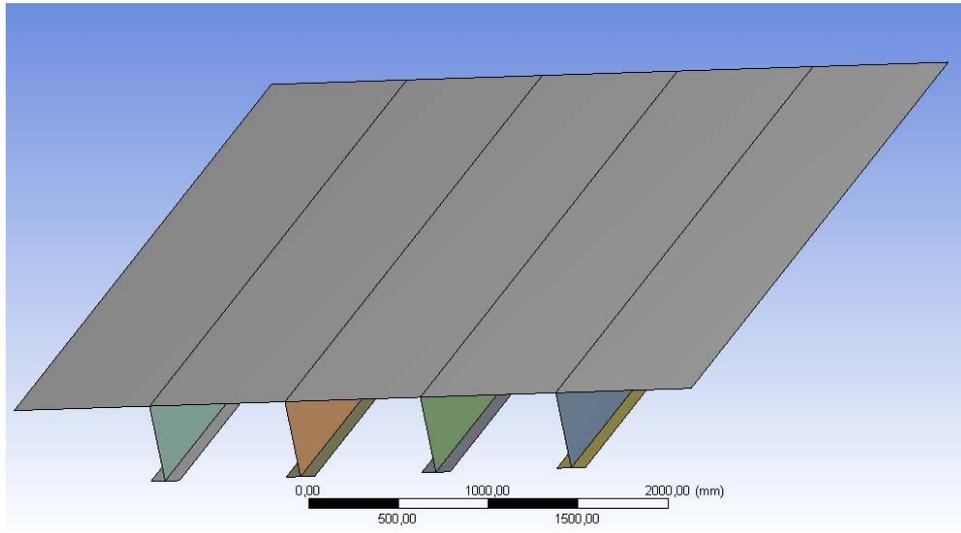


Figure 9.7: The geometrical model

Both, plate and stiffeners are made of structural steel, with the following bilinear elastic characteristics:

- Yield stress:  $\sigma_y = 315 \text{ MPa}$
- Strain at yield  $\epsilon_y = 0.001531$
- Young's Modulus:  $E = 205.8 \text{ GPa}$
- Tangent Modulus:  $E_t = 1000 \text{ MPa}$

The stress-strain curve of the used steel is depicted in Figure 9.8.

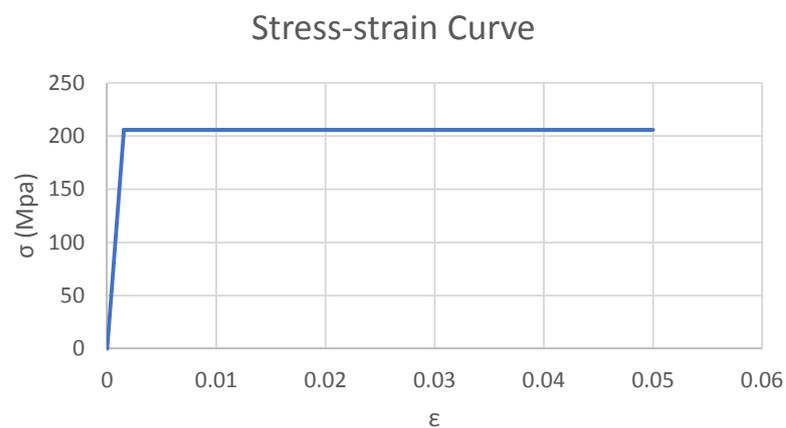


Figure 9.8: The stress-strain curve

The solution process is depicted in Figure 9.9:

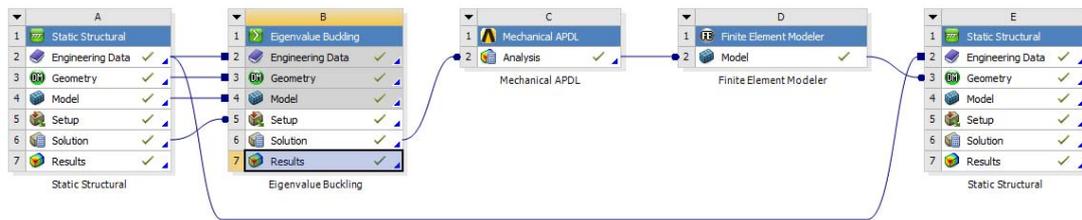


Figure 9.9: The project schematic

In order to prepare the deformed shape which will be used as initial imperfection, the linear buckling solution is obtained. The first linear eigenvalue buckling mode is shown in Figures 9.10a and 9.10b.

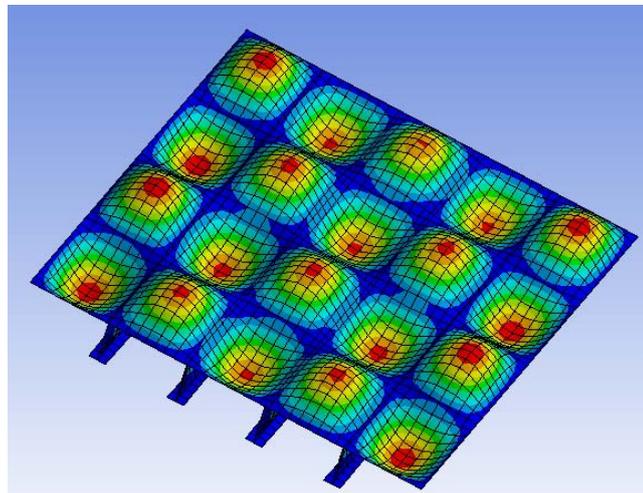


Figure 9.10a: The first linear eigenvalue buckling mode – Plate view

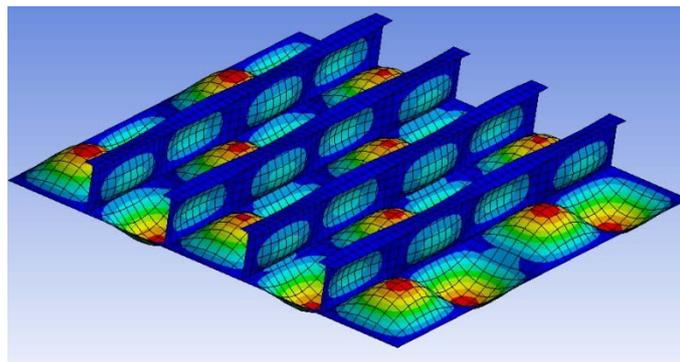


Figure 9.10b: The first linear eigenvalue buckling mode – Stiffener view

Next, the UPGEOM command of the ANSYS is used in order to prepare the “deformed” model according to the first eigenvalue mode. Then the model is introduced to a new Static Structural Problem where the non-linear solution is obtained.

In Figures 9.11a and 9.11b the buckled structure, after the non-linear solution is indicated.

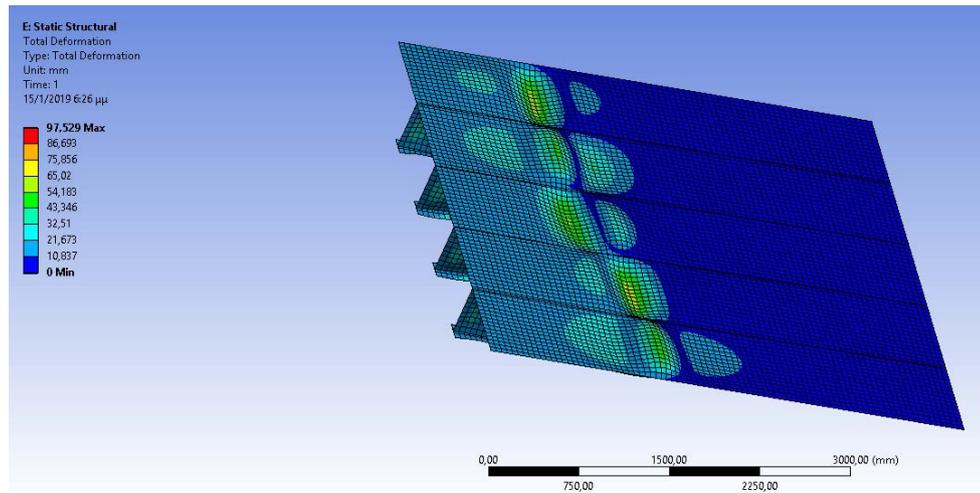


Figure 9.11a: The buckled structure – Plate view

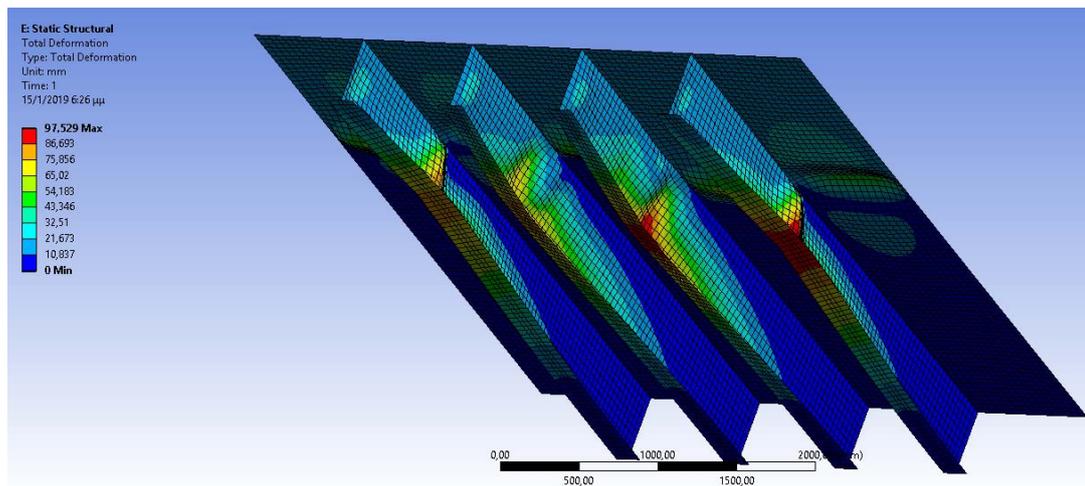


Figure 9.11b: The buckled structure – Stiffener view

In Figure 9.12 the normalized resulting Load Displacement curve is shown. From the shape of the curve it follows that the collapse of the panel occurs when the average normal compressive stress along the short edge of the stiffened plate approaches the value:

$$\sigma_{\text{collapse}} = 0,9 \times \sigma_y = 185 \text{ MPa.}$$

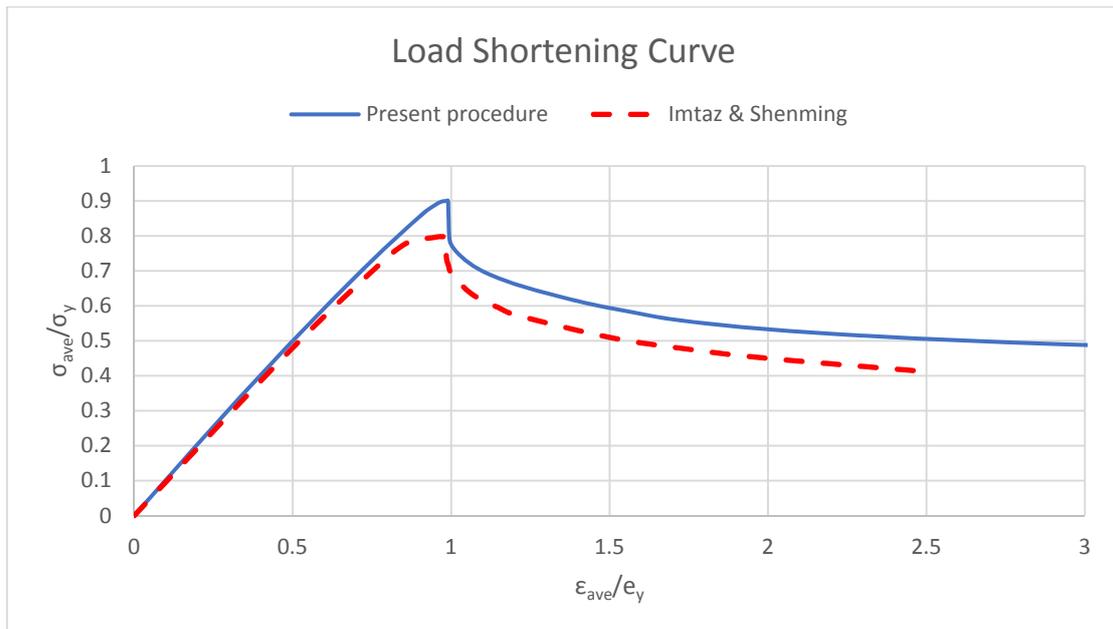


Figure 9.12: The normalized load shortening curve

The comparison with the referenced publication is considered satisfactory, given that different software, different initial imperfections and different iterative scheme (Static Rics) has been used.

## 10. References

1. ANSYS Workbench User's Guide.
2. Madenci, E., Guven, I., "The Finite Element Method and Applications in Engineering Using ANSYS", Springer International Publishing, 2015.
3. "Non-linear Structural Collapse Analysis for Plates and Stiffened Panels", ShipRight Design and Construction- Additional Design Procedures, May 2016.
4. Imtaz Khan, Shengming Zhang, "Effects of welding-induced residual stress on ultimate strength of plates and stiffened panels", Ships and Offshore Structures, Vol. 6, No. 4, pp 297-309, 2011.