SESAM EXAMPLE

Linearized Buckling, P-Delta and Stress Stiffening Analyses of Jackup Leg

Sesam Example

Linearized Buckling, P-Delta and Stress Stiffening Analyses of Jackup Leg

Date: April 2023

Prepared by: Digital Solutions at DNV

E-mail support: software.support@dnv.com

E-mail sales: digital@dnv.com

Table of contents

1 Introduction

This document is about the three analysis types: linearized buckling, P-Δ (P-delta) and stress stiffening. The example is named JackupLeg_LinBck_Pdelta_StrStiff and is run as a job in Sesam Manager. See **Figure 1-2** below.

The model shown in **Figure 1-1** below is a single jackup leg. The jackup deck is simplified to a triangular prism. The plates of the prism have (unrealistically) big thickness and high density so as to represent a third of the deck mass of a three-legged jackup. The triangular prism is given boundary conditions with fixed rotations so as to mimic the deformation pattern of jackup legs enforced by the deck as illustrated below.

The example is based on units kN and m. Acceleration of gravity is set to 9.80665. The following programs and versions are used: GeniE 8.5, Wajac 7.9, Sestra 10.17, and Xtract 6.0.

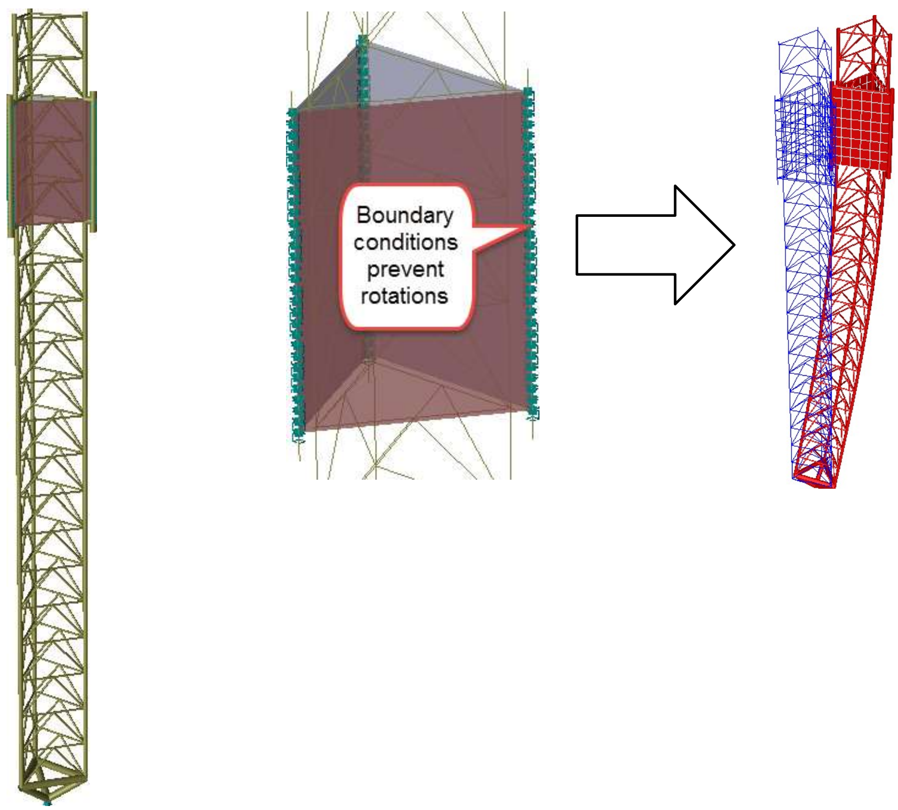

Figure 1-1 Jackup leg for linearized buckling, P-Δ and stress stiffening analyses

- JackupLeg_LinBck_Pdelta_StrStiff
  - LinearBuckling
    - GeniELinearBucklingModel
    - SestraLinearBucklingAnalysis
    - XtractVerifyBucklingMode
  - Pdelta
    - GeniEPdeltaModel
    - SestraPdeltaAnalysis
    - XtractPdelta
  - StressStiffening
    - GeniEStressStiffeningModel
    - SestraStressStiffeningAnalysis
    - XtractStressStiffening

Figure 1-2 Sesam Manager workflow shown as 'Tree View'

2 Linearized Buckling Analysis

Sestra offers linearized buckling analysis as described in the Sestra user manual. Linearized buckling analysis is triggered by setting the LBUCK parameter as 1 on the CMAS command. The EIGA command demanding computation of eigenvalues (of which only the first is normally of interest) and with a shift value is also required.

A linearized buckling analysis is about determining the factor to be applied to a given load distribution so that buckling will occur. The factor is termed ‘stability factor’ or ‘critical load level’. The functionality in Sestra is available for beam and plate/shell structures.

Linearized buckling analysis is a two-step procedure automatically performed by Sestra:

1. Static analysis is performed for the load for which the stability factor is desired.
    - **K** is the ordinary linear stiffness matrix.
    - Geometric stiffness matrix $\mathbf{K}_g$ is found based on stresses in plates/shells and forces in beams.
2. Buckling modes **Φ** and corresponding eigenvalues λ are found by solving the eigenvalue problem: $(\mathbf{K} - \lambda\mathbf{K}_g)\mathbf{\Phi} = \mathbf{0}$
    - The eigenvalues λ are the sought stability factors.
    - Only the first eigenvalue (buckling mode) and the corresponding stability factor are of interest.

A linearized buckling analysis may be used to find the buckling load P_E used in a P-Δ analysis. The stability factor corresponds to the ratio P_E/P in the formula for amplification factor α, see section 3.

Setting up a linearized buckling analysis is done as follows:

- Ensure that the load causing compression is the last load case or load combination. Alternatively, the command GSTF can be used to select the buckling load.
- Set parameter LBUCK on command CMAS equal to 1.
- Add command EIGA with an appropriate shift value.

Example of Sestra input for a linearized buckling analysis:

```
COMM   CHCK  ANTP  MSUM  MOLO  STIF  RTOP  LBCK
CMAS     0.    1.    1.    0.    0.    0.    1.
ITOP     1.
COMM    ENR                                                            SHIFT
EIGA     2.                                                            0.1
Z
```

In the present analysis a stability factor of 1.566 is found (in the sestra.lis file) for a combination of gravity and buoyancy. I.e. the frame structure will collapse when gravity minus buoyancy is multiplied by 1.566.

3 P-Δ Analysis

The P-Δ effect is a 2nd order effect: A horizontal force H gives moment M_H and horizontal displacement Δ. The vertical force P moves with the horizontal displacement. This causes an additional moment PΔ and an additional horizontal displacement.

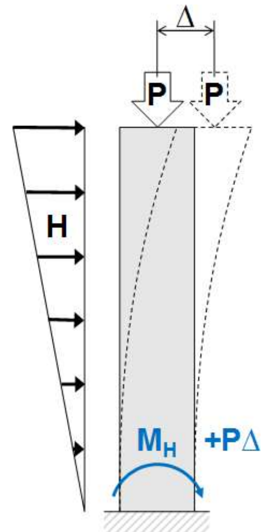

Figure 3-1 Horizontal force gives horizontal displacement that combined with vertical force produces additional moment

A proposal for a P-Δ analysis is as follows:

- For a vertical structure with uniform cross-sectional properties over the height – in essence a beam – compute the buckling load P_E:
  - $P_E = \pi^2 EI/(kl)^2$
  - The section modulus I may be found by hand calculation.
  - The buckling factor k depends on the boundary conditions of the beam as shown in **Figure 3-2** below. The figure to the left with k=2 is the one corresponding to the present structure.

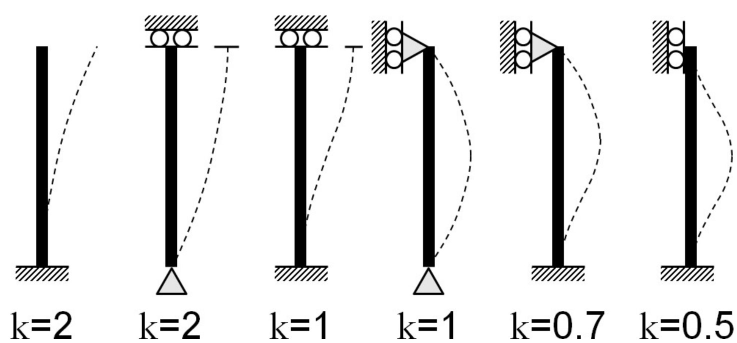

Figure 3-2 Buckling factors depending on boundary conditions

  - Find the horizontal displacement amplification factor $\alpha = 1/(1–P/P_E)$.
- For structures that cannot be regarded as beams with uniform sectional properties over the height a linearized buckling analysis, see section 2, may be performed. In this case the stability factor corresponds to the ratio P_E/P.

- A static linear analysis is performed with wave, current, wind and other horizontal loads to find the horizontal displacement Δ at top.
- Apply a horizontal load (scale an arbitrary load) at top to produce an additional horizontal displacement (α–1)Δ – the total horizontal displacement is then αΔ.

In the present case a linearized buckling analysis has been performed, see section 2, and the stability factor has been found to be 1.566. This gives amplification factor α = 1/(1–1/1.566) = 2.767.

Two wave loads are analyzed, one propagating in direction 0 degrees and another in 45 degrees. Correspondingly, two auxiliary horizontal loads are defined as shown in **Figure 3-3** below, one in X direction (0 degrees) and another in X and Y directions combined (45 degrees). By viewing the results in Xtract the horizontal displacements in a selected node at top of the triangular prism, see the figure to the right below, are found and used to compute the scaling factors. This node at top of the triangular prism is selected assuming that this is the elevation of most of the gravity being the P in the P-Δ analysis.

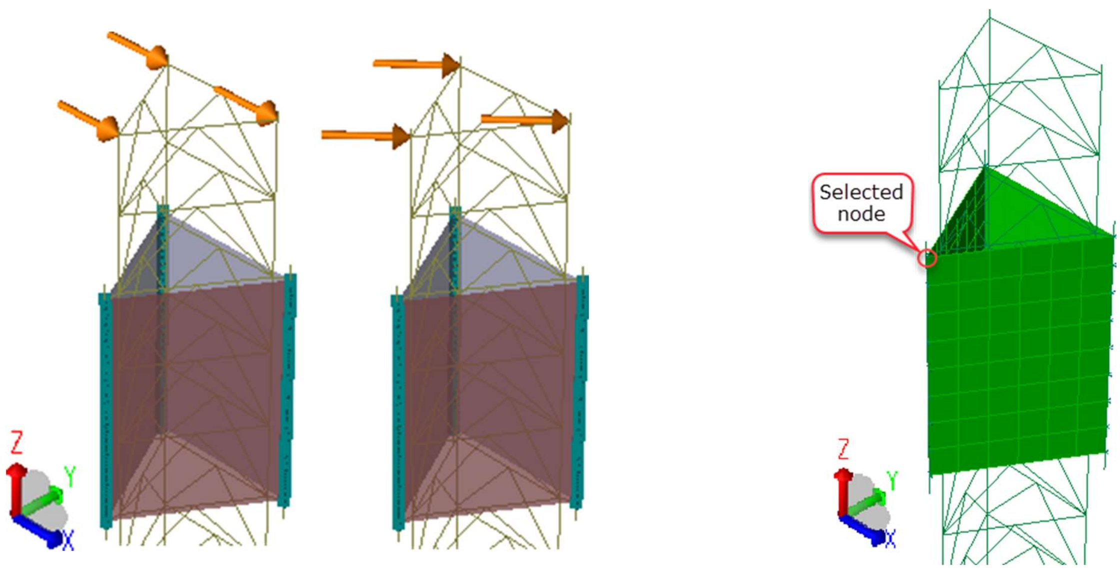

Figure 3-3 Auxiliary horizontal forces and selected node for finding horizontal displacements

For 0-degree direction:

- The auxiliary load gives displacement (component X) 9.631E–7, see the upper left figure in **Figure 3-4** below.
- Gravity + wave (WLC1) + buoyancy (WLC3) (Xtract load combination LoadIn0_Grav_Wave_Buoy) gives displacement (component X) Δ = 0.09763, see the upper right figure below.
- This means that the scaling factors for the load is: $(\alpha–1)\Delta/9.631E–7 = (2.767–1)*0.09763/9.631E–7 = 179100$.
- A load combination (LoadIn0_WithPdelta) is made in Xtract by adding the auxiliary load times 179100 to the wave + gravity + buoyancy load. This combination is used in further calculations.
- The combination gives displacement (component X) = 0.2701, as seen in the lower figure below.

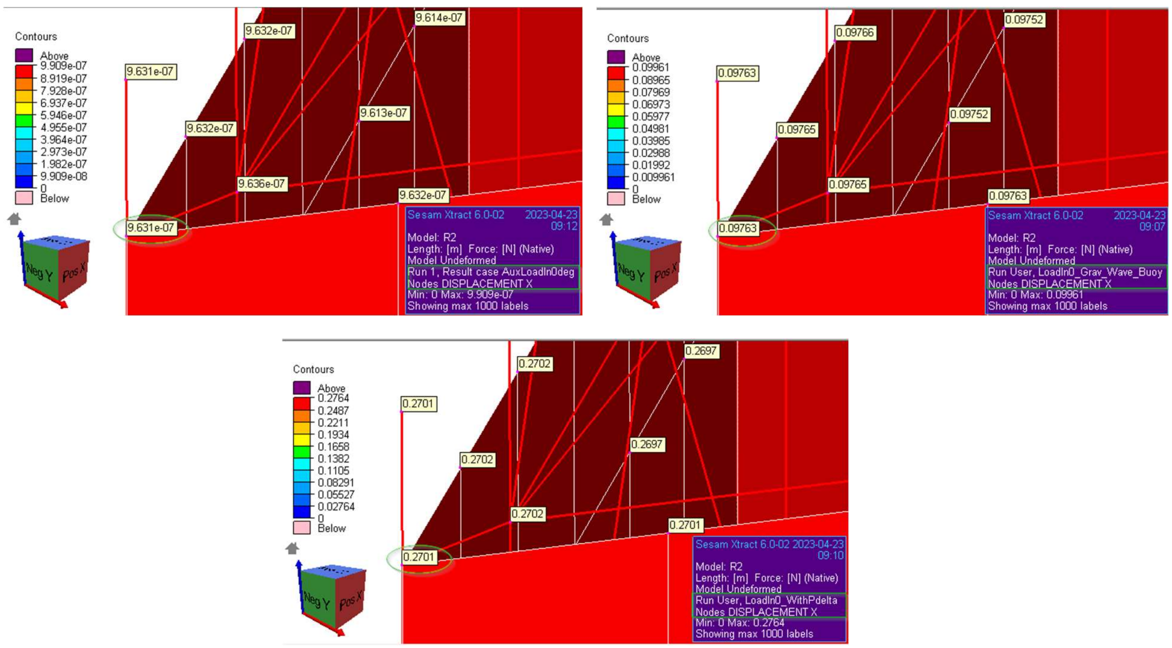

Figure 3-4 X-displacements for auxiliary horizontal forces and two load combinations

For 45-degree direction:

- The horizontal load gives displacement (components X and Y combined), see the upper left figure in **Figure 3-5** below:
  $\sqrt{(9.631E\text{-}7)^2 + (9.635E\text{-}7)^2} = 1.362E{-}6$
- Gravity + wave (WLC2) + buoyancy (WLC3) (Xtract load combination LoadIn45_Grav_Wave_Buoy) gives displacement (components X and Y combined), see the upper right figure below:
  $\Delta = \sqrt{(0.06896)^2 + (0.06902)^2} = 0.09757$
- This means that the scaling factors for the load is:
  $(\alpha{-}1)\Delta/1.362E{-}6 = (2.767{-}1)*0.09757/1.362E{-}6 = 126600$
- A load combination (LoadIn45_WithPdelta) is made in Xtract by adding the horizontal load times 126600 to the wave load. This combination rather than the wave load is used in further calculations.
- The combination gives displacement (components X and Y combined), as seen in the lower figure below:
  $\sqrt{(0.1909)^2 + (0.1910)^2} = 0.2700$

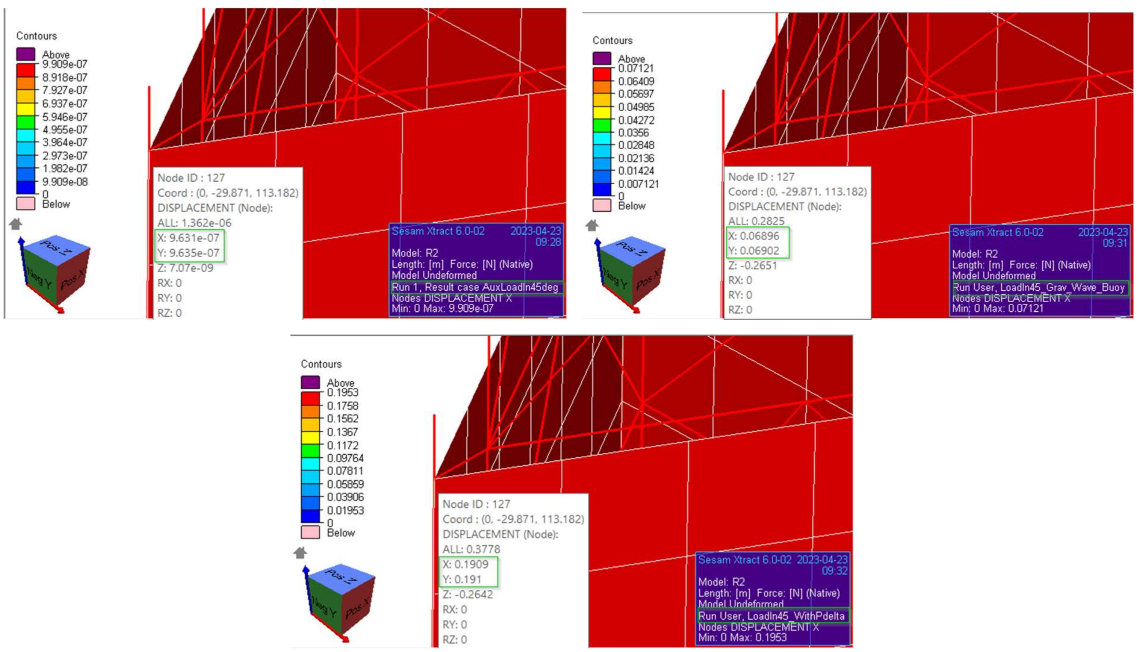

Figure 3-5 X- and Y-displacements for auxiliary horizontal forces and two load combinations

4 Stress Stiffening Analysis

Sestra offers stress stiffening analysis as described in the Sestra user manual. In such an analysis the stiffening effect of tensile stresses/forces and softening effect of compressive stresses/forces are accounted for. (A more descriptive name of the method would thus be ‘stress stiffening/softening analysis’.) The functionality in Sestra is available for beam and plate/shell structures.

Stress stiffening analysis is a two-step procedure automatically performed by Sestra:

1. Static analysis of load (typically gravity, buoyancy and other loads with vertical components) is performed to find initial stresses in plates/shells and forces in beams. In the present analysis the gravity and buoyancy are included in the stress stiffening case. The contribution to vertical loading from the wave load is relatively small and therefore excluded from the stress stiffening case. This is convenient as it means that the same stress stiffening case is used for wave loading from any direction.
    - Geometric stiffness (or initial stress) matrices of the basic elements are calculated based on initial stresses.
2. Geometric stiffness matrices are added to the ordinary stiffness matrices of the basic elements.
    - An updated stiffness matrix for the model is achieved and based on this a new analysis is performed – this may be a static, free vibration or forced dynamic analysis.

A stress stiffening analysis may be highly relevant in an eigenvalue analysis since the softening of a slender structure due to compressive stresses and forces may significantly increase the eigenperiods (natural periods).

Setting up a stress stiffening analysis is done as follows:

- Ensure that the load causing tension/compression is the last load case or load combination. Alternatively, the command GSTF can be used to select the stress stiffening load.
- Set parameter STIF on command CMAS equal to 2.

Example of Sestra input for a static analysis with stress stiffening:

```
COMM  CHCK ANTP MSUM MOLO STIF
CMAS    0.   1.   1.   0.   2.
ITOP    3.
Z
```

Example of Sestra input for an eigenvalue analysis computing 10 modes with stress stiffening:

```
COMM CHCK ANTP MSUM MOLO STIF
CMAS   0.   2.   1.   0.   2.
ITOP   3.
EIGA  10.
IDTY   3.
DYMA   1.
Z
```

In the present analysis a static analysis of waves plus current from two directions (0 deg and 45 deg), gravity and buoyancy shall be performed with the gravity plus buoyancy as stress stiffening case. Therefore, a combination (StressStiffCase) of the gravity load case and the buoyancy load case is made. Smart load combinations must be switched off in GeniE (edit the meshing activity) to make the load combination available to Sestra. The load causing tension/compression will then be the last load case as required by a stress stiffening analysis (unless command GSTF is used).

In the present analysis the displacements of the selected node at top of the triangular prism are:

- For 0-degree direction (component X): 0.2701 which is exactly the same value as for the P-delta analysis.
- For 45-degree direction (components X and Y combined):
  $\sqrt{((0.1908)^2 + (0.1907)^2)} = 0.2698$ – the P-delta analysis gave 0.2700

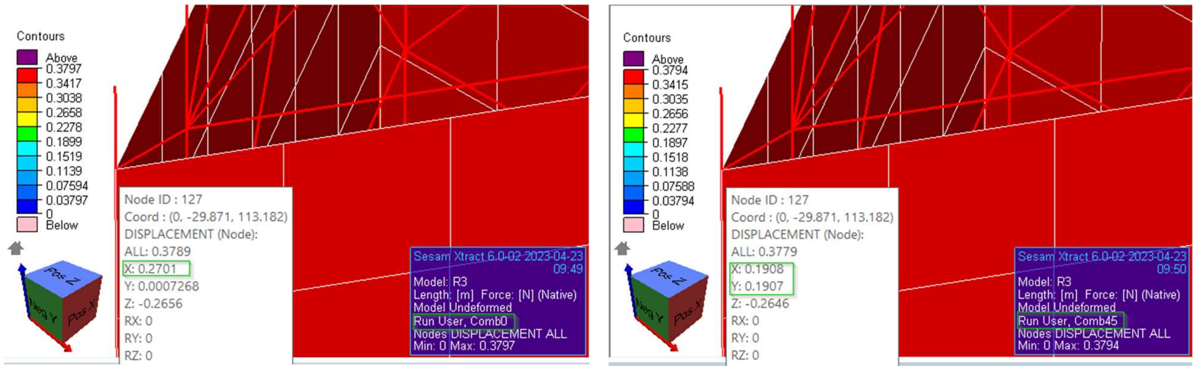

Figure 4-1 X- and Y-displacements for wave+current, gravity and buoyancy combined in directions 0 and 45

About DNV

We are the independent expert in risk management and quality assurance. Driven by our purpose, to safeguard life, property and the environment, we empower our customers and their stakeholders with facts and reliable insights so that critical decisions can be made with confidence. As a trusted voice for many of the world's most successful organizations, we use our knowledge to advance safety and performance, set industry benchmarks, and inspire and invent solutions to tackle global transformations.

Digital Solutions

DNV is a world-leading provider of digital solutions and software applications with focus on the energy, maritime and healthcare markets. Our solutions are used worldwide to manage risk and performance for wind turbines, electric grids, pipelines, processing plants, offshore structures, ships, and more. Supported by our domain knowledge and Veracity assurance platform, we enable companies to digitize and manage business critical activities in a sustainable, cost-efficient, safe and secure way.

DNV | digital@dnv.com | dnv.com/digital