COMPASS
www.compassis.com

SeaFEM - Validation Case 11

Cable under self-weight

SeaFEM

Version
15.1.0

Compass Ingeniería y Sistemas http://www.compassis.com
Tel.: +34 932 181 989 - Fax.: +34 933 969 746 - E: info@compassis.com - C/ Tuset 8, 7o 2a, Barcelona 08006 (Spain)

http://www.compassis.com
info@compassis.com
November 2018

Table of Contents

1 Problem description

This test case aims to validate the dynamic cable model implemented in SeaFEM. This simple model consists on a cable fixed at both ends and subjected to the action of its self-weight. The solution of the dynamic cable model can be compared against the analytical solution of an elastic catenary with the same length. The asimptotic solution of the dynamic cable model should provide the same deflection as the elastic catenary solution. Additionally, the vertical reaction force at both ends of the cable can serve also as validation since it must be equal to half the total weight of the cable.

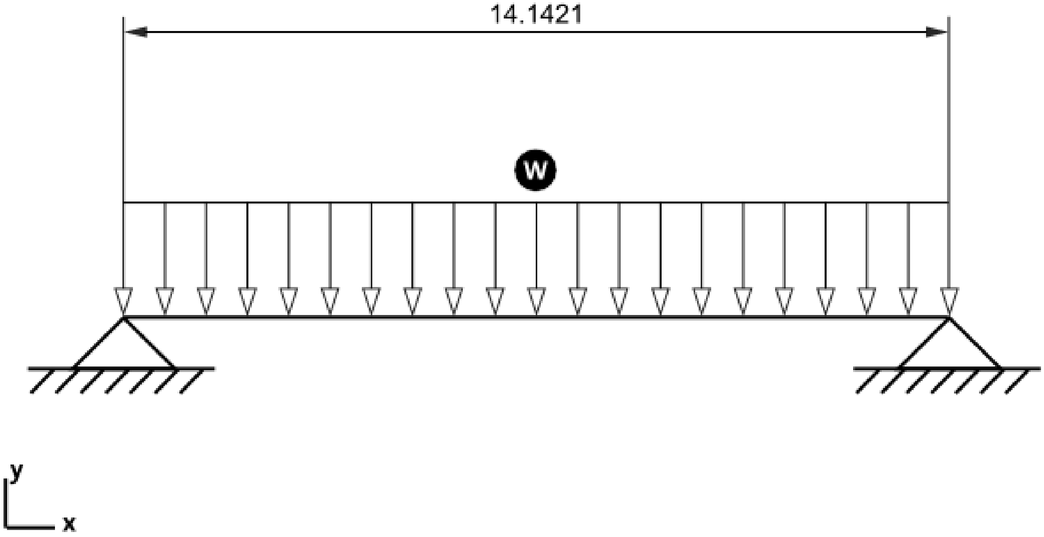
Following Ref. [1], the data used for this test is:

A=0.0005 m^2 (Area of the cable element section).

E=5.01E6 N/m^2 (Young modulus of the material).

L=14.1421 m (Length of the cable).

w=0.49 N/m (Effective weight per unit length).

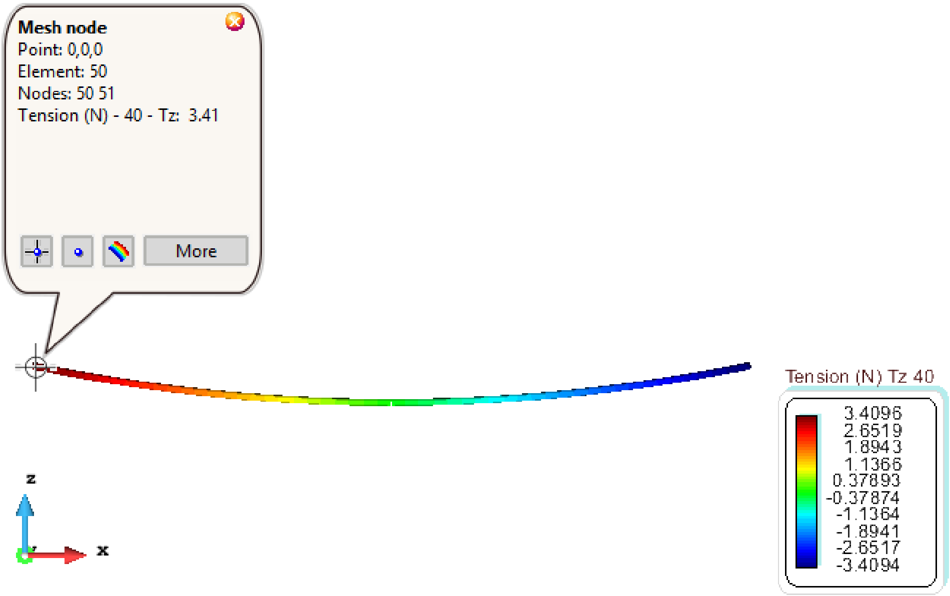


Geometric description of thte test model.

2 Results

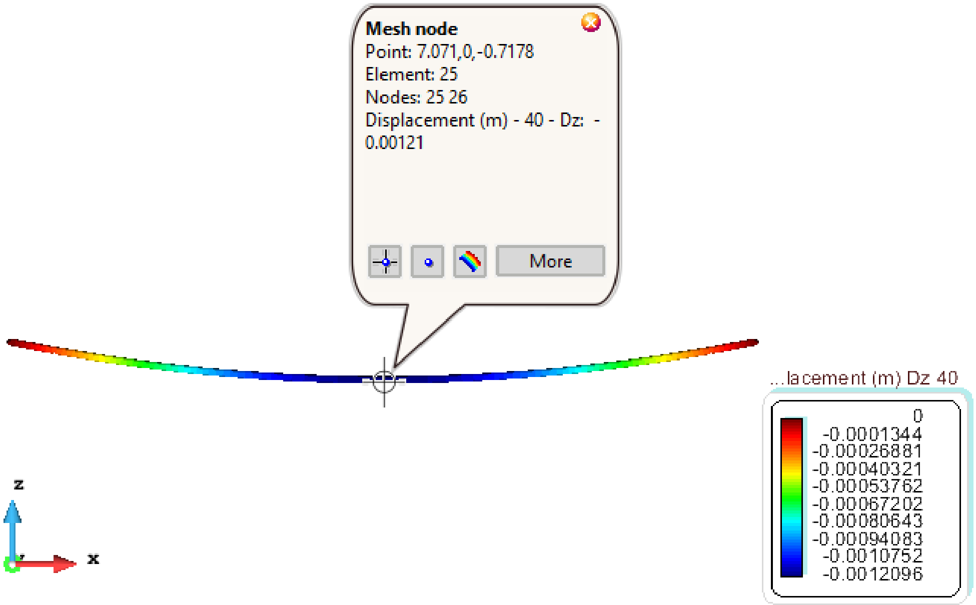
In order to assess numerical results precission and model validation, the results from the simulation are compared against the analytical solution corresponding to a catenary of length L with the same deflection. Additionally, vertical reaction results are also compared.

Given the properties of the cable considered for the present analysis, the total weight of the cable is W = 6.93 N, so that the vertical reactions at the fixed ends of the cable in the equilibrium position must be R = 3.465 N. As can be seen in the following figure, where the vertical component of the tension at one end of the cable is shown, the assimptotic value of the vertical reaction reached at the equilibrium position of the cable is about R = 3.410 N which is in good agreement with the expected value.



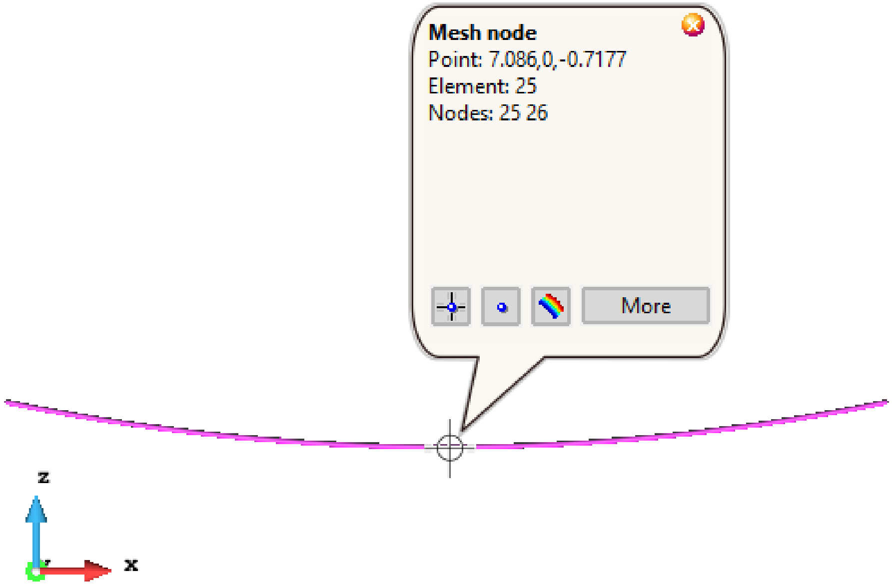
Vertical component of the cable's tension at one end of the cable

On the other hand, in the following figure we present the displacement results of the node where maximum deflection of the cable occurs. As it can be expected, maximum deflection occurs at the center of the cable.



Vertical displacement at the central point of the cable

It must be noted that displacement results reported by SeaFEM are referred to the initial reference configuration which is obtained at the beginning of the simulation by solving the static catenary problem. Hence, to obtain the total displacement of the central point of the cable we must add to the reported displacement value, the vertical displacement of the initial reference configuration. To this aim, the coordinates of the cable's central point at the initial configuration must be obtained, using the post-process mesh information tools, as shown in the following figure.



Initial reference configuration coordinates of the cable's central point

Hence, the total displacement of the point under analysis is:

dz_max = -0.7177 -0.00121 = -0.7189 m.

Such maximum deflection of the cable can be validated against the value corresponding to the solution for a catenary of the same final length. Such a solution predicts that the maximum deflection is given by:

$$fl = \frac{wL^2}{8T_h}$$

being T_h the horizontal tension of the catenary that can be obtained by solving numerically the static equations of the catenary [2]. This expression results in a value for the maximum deflection of fl = 0.72 m., which is in very good agreement with the value fl = -0.7189 m. obtained from the finite elements results of the dynamic cable model presented above.

3 Appendix

The following code describes the tcl script used for the present analysis. `TdynTcl_CreateMooring` procedure is used to create the dynamic cable segment and to specify its configuration. It can be used as well to enforce the displacements at the end of the cable if necessary.

Note that in newer versions of the program mooring lines can be already defined using the graphic user interface of SeaFEM.

```
proc TdynTcl_InitiateProblem { } {
    configure_analysis Solve_Dif_Rad 0
}

proc TdynTcl_CreateMooring { } {
    # Validation Case 2
    # Time step suggested for simulation: 0.01 sec.

    # values: Body Type_Mooring Xe[3] Xi [3] w L A E S n_elements damp_ratio_1 
damp_ratio_2
    set seg1  [create_mooring_segment 1 6 0.0 0.0 0.0 14.1421 0.0 0.0 0.49 
14.1421 0.0005 5.01e6 0 22 0.15 0.15]
    # values: Gk Gc Gu Ms Md Cd Cf Cm alpha_bs gamma_bs bci steps
    TdynTcl_Configure_Mooring_Segment 1 0.0 0.0 0.0 0.0 0.0 0.0 0.0 0.0 -0.1 
0.9 1 1

    set fun1 [::mather::create_function waves "0.0;"]
    set fun2 [::mather::create_function waves "0.0;"]
    set fun3 [::mather::create_function waves "0.0;"]
    TdynTcl_Set_Mooring_Displacement $seg1 $fun1 $fun2 $fun3 0
}
```

4 Validation Summary

CompassFEM version	15.1.0
Tdyn solver version	15.1.0
RamSeries solver version	15.1.0
Benchmark status	Successfull
Last validation date	27/11/2018

5 References

[1] Ortigosa, I. Development of a decision support system for the design and adjustment of sailboat rigging. PhD Thesis, Universitat Politècnica de Catalunya. 2011.

[2] Gutiérrez, J.E. Desarrollo de herramientas software para el análisis de aerogeneradores offshore sometidos a cargas acopladas de viento y oleaje. PhD Thesis, Universidad Politécnica de Cartagena. 2014.