Manual

for the

Ansys Exercise

Accompanying the

Offshore Wind Farm Design

Assignment

Contents

1. Introduction

This manual is intended to assist students in using the 'ANSYS Macros for Offshore Wind Turbines' for the assignment accompanying the Offshore Wind Farm Design course. The manual will address the following points:

- Brief overview of the finite element program ANSYS
- Overview of the input files for the ANSYS Macros
- Getting started with the macros
- Performing a modal analysis
- Determining the pile penetration depth
- Determining the translational and rotational ground springs required in BLADED

2. Brief Overview of ANSYS

A professional finite element tool such as ANSYS is a sophisticated piece of software with many possibilities. This manual will touch only briefly on the most basic issues in order to clarify the general sequence of operations to arrive at an end result.

Figure 1 shows the Graphic User Interface (GUI) of the program. The window to the left, in the red oval, displays the *Main menu*. In this menu three menu items are important, the *Preprocessor*, the *Solution* phase and the *General postprocessor*.
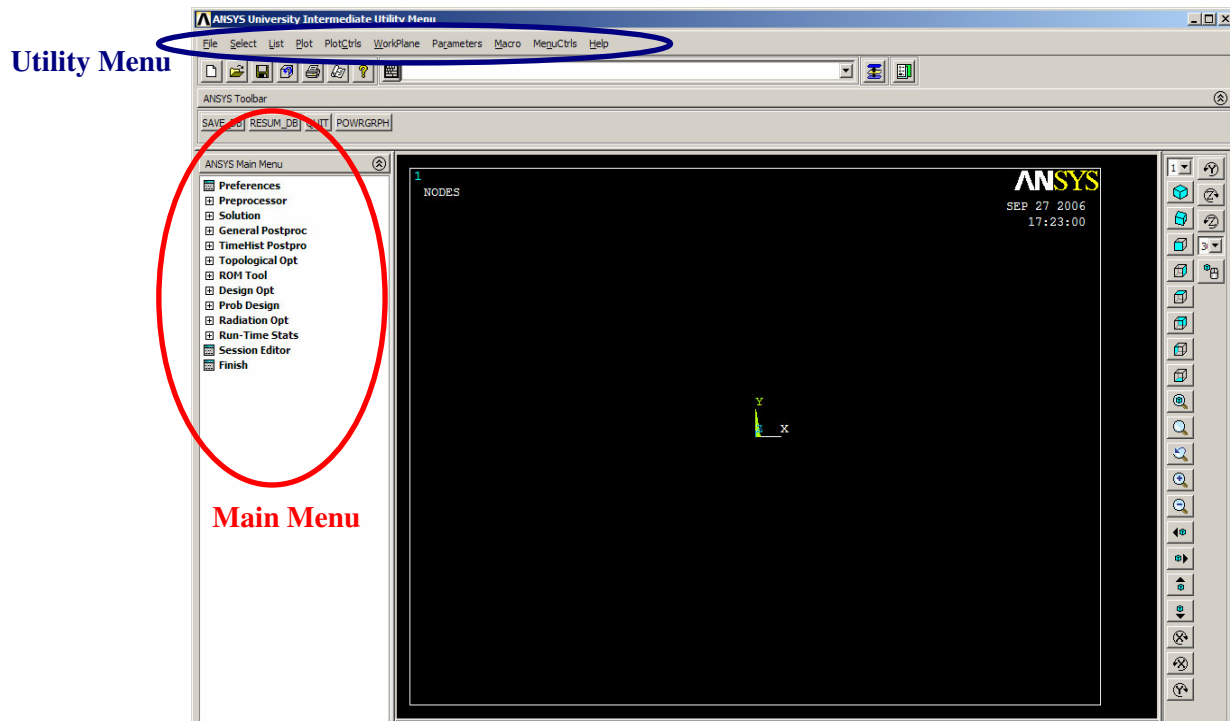
Figure 1: *Ansys Graphic User Interface*

In the *Preprocessor*, a model can be built and modified. Loads and constraints can also be applied. In the *Solution* phase, the type of analysis to be performed (static, modal, transient, etc) can be specified. The results of an analysis can be viewed in the *General postprocessor*. Several useful commands can be accessed via pull-down menus in the *Utility menu*, marked by the blue oval. On the right hand side of the screen, several buttons can be found that allow the user to adjust the view.

3. Overview of the input files for the ANSYS Macros

Figure 2. Example of an offshore wind turbine support structure

3.1   Struct.dat: Structure file

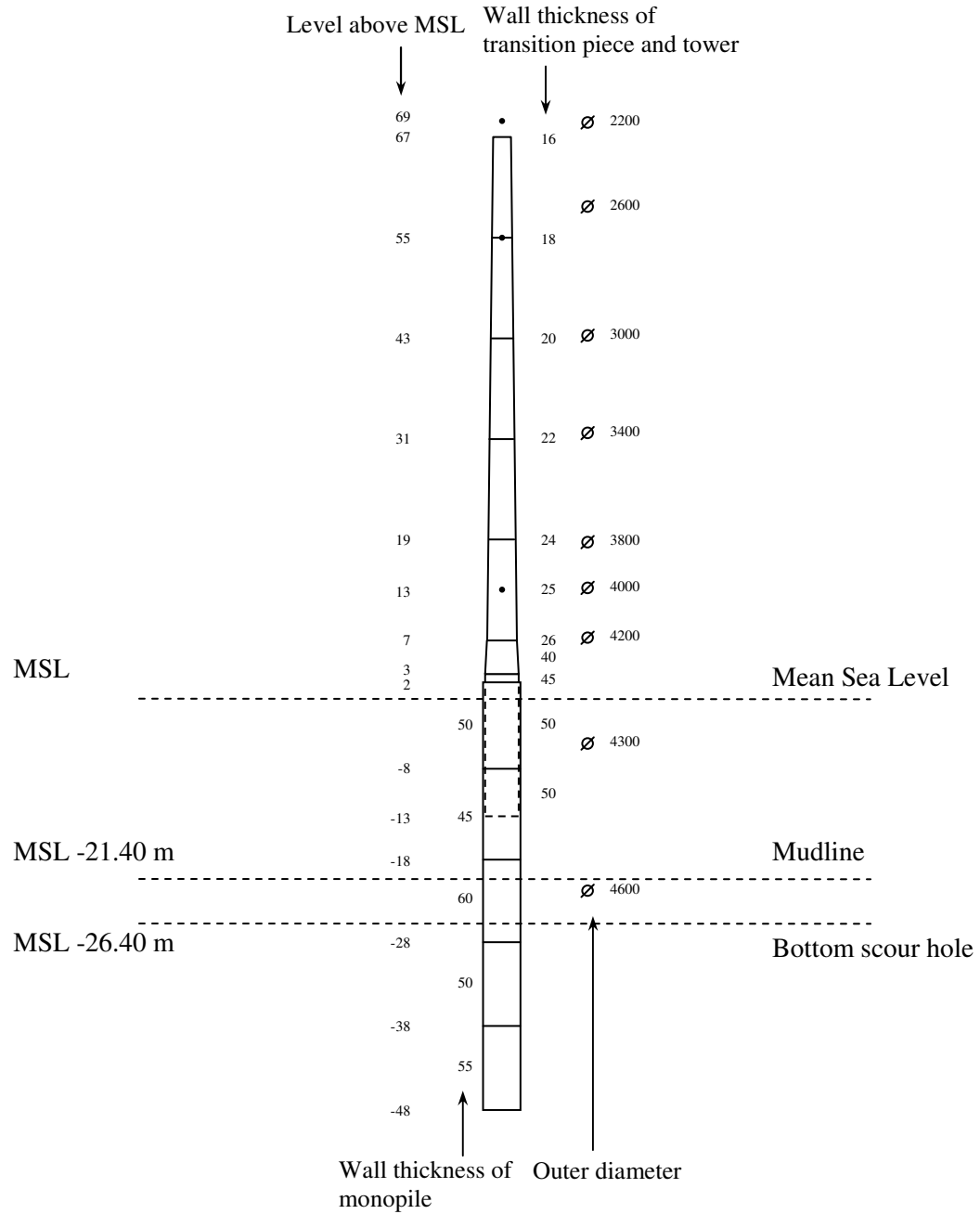This file contains the geometry of the support structure. The diameter, wall thickness and concentrated masses can be entered for each desired elevation. This file will be explained with the example given in figure 2 in mind.

In this example the transition piece fits inside the monopile. The diameter of the monopile is 4.6 m and the diameter of the transition piece is 4.3 m. The annulus is filled with grout. The corresponding structure file is given below. Concentrated masses are defined at 13 m and 55 m above MSL, while the centre of gravity of the nacelle is located at 69 m above MSL.

```
! COORDINATE SYSTEM WITH RESPECT TO MSL
offset(1)    =        0,0,0

! PILE
pile(1,1)    =        4.600    ,        steel
pile(1,2)    =        -48      ,        0.055
pile(1,3)    =        -38      ,        0.050
pile(1,4)    =        -28      ,        0.060
pile(1,6)    =        -18      ,        0.045
pile(1,5)    =        -13      ,        0.000            !        END OF PILE

! TUBULAR
tubular(1,1,1)    =        -13    ,        4.300    ,        0.045    ,        steel
tubular(1,2,1)    =        -8     ,        4.300    ,        0.045
tubular(1,3,1)    =        -8     ,        4.300    ,        0.050    ,        steel
tubular(1,4,1)    =        2      ,        4.300    ,        0.050

tubular(1,1,2)    =        -13    ,        4.300    ,        0.050    ,        steel
tubular(1,2,2)    =        -8     ,        4.300    ,        0.050
tubular(1,3,2)    =        -8     ,        4.300    ,        0.050    ,        steel
tubular(1,4,2)    =        2      ,        4.300    ,        0.050
tubular(1,5,2)    =        2      ,        4.300    ,        0.045    ,        steel
tubular(1,6,2)    =        3      ,        4.300    ,        0.045
tubular(1,7,2)    =        3      ,        4.300    ,        0.040    ,        steel
tubular(1,8,2)    =        7      ,        4.200    ,        0.040
tubular(1,9,2)    =        7      ,        4.200    ,        0.026    ,        steel
tubular(1,10,2)   =        13     ,        4.000    ,        0.025    ,        steel    , , 20000
tubular(1,11,2)   =        19     ,        3.800    ,        0.024    ,        steel
tubular(1,12,2)   =        31     ,        3.400    ,        0.022    ,        steel
tubular(1,13,2)   =        43     ,        3.000    ,        0.020    ,        steel
tubular(1,15,2)   =        55     ,        2.600    ,        0.018    ,        steel    , , 7000
tubular(1,16,2)   =        67     ,        2.200    ,        0.016    ,        steel

! TOWER TOP
nacelle(1,1)    =        0.00    ,        0.000    ,        70.000
nacelle(1,2)    =        120000
nacelle(1,3)    =        0       ,        0        ,        0

/EOF
```

The structure file consists of 4 sections. The first section allows the user to define an offset of the origin of the structure with respect to the reference level (i.e. MSL or LAT). The second section contains the data for the monopile. The first line in this section requires the outer diameter of the monopile. This value is assumed to be constant over the entire length of the monopile. It is not possible to incorporate variations of the outer diameter of the monopile in the TRF Tool. The material used is also required.

```
pile(1,1)    =            4.600     ,          steel

                     pile diameter       material
```

In the subsequent lines the elevation at the bottom of the segment and the wall thickness of the segment is required, starting with the bottommost segment.

pile(1,2)   =   -48   ,   0.055

     elevation at lower   wall thickness
     end of segment   of segment

The last line in the section gives the elevation at the top of the topmost section. The wall thickness in this line should set to 0.

pile(1,5)   =   -13   ,   0.000

     elevation at upper   wall thickness
     end of top segment must be set to 0

Although figure 5 shows that the top of the monopile is at 2 m above MSL, in the structure file it ends at 13 m below MSL. This is where the transition piece starts. The reason for this is the way the macros in Ansys deal with an overlap. This is explained in the following.

The second section gives the geometry of the transition piece and the tower. In contrast to the previous section this section allows variations of the outer diameter and wall thickness. If there is a discontinuity with respect to diameter or wall thickness the level at which this discontinuity occurs should be given twice, once for the values below the discontinuity and once for the values above it.

tubular(1,2,1)   =   -8   ,   4.300   ,   0.045
tubular(1,3,1)   =   -8   ,   4.300   ,   0.050   ,   steel

           Elevation of   Diameter   Discontinuity of wall
           discontinuity   thickness

As was mentioned earlier an overlap can be modeled. In this case the monopile is on the outside and the transition piece is on the inside. The portion that connects to the lower part of the structure should be given first. The final index of this potion should be 1. Subsequently, the portion that connects to the upper part of the structure should be given. The final index of this potion should be 2. A Part of the structure file is given below to illustrate this. It should be noted that the elevations at which any properties are defined for the overlap should coincide for both the inner element and the outer element. If the approach as described above is followed, the macros instruct Ansys to fill the annulus with grout.

Index should be 1   Elevations should coincide   Material given at beginning of segment

tubular(1,1,1)   =   -13   ,   4.300   ,   0.045   ,   steel
tubular(1,2,1)   =   -8   ,   4.300   ,   0.045
tubular(1,3,1)   =   -8   ,   4.300   ,   0.050   ,   steel
tubular(1,4,1)   =   2   ,   4.300   ,   0.050

tubular(1,1,2)   =   -13   ,   4.300   ,   0.050   ,   steel
tubular(1,2,2)   =   -8   ,   4.300   ,   0.050
tubular(1,3,2)   =   -8   ,   4.300   ,   0.050   ,   steel
tubular(1,4,2)   =   2   ,   4.300   ,   0.050

Index should be 2

In the case of segments with continuously varying properties, the diameter and wall thickness along with the elevation must be given at the beginning and the end of the segment. If the properties at the end of the segment are equal to the properties at the beginning of the next segment, there is no need to repeat these properties. The material should be mentioned at the beginning of each segment. A concentrated mass can be added at any desired level as shown below.

<div style="color:red">Material defined at
beginning of segment</div>

```
tubular(1,9,2)    =      7    ,    4.200   ,   0.026   ,   steel
tubular(1,10,2)   =      13   ,    4.000   ,   0.025   ,   steel   ,  , 20000
tubular(1,11,2)   =      19   ,    3.800   ,   0.024   ,   steel
```

<div style="color:blue">Properties given once
per elevation</div>

<div style="color:blue">Concentrated
mass added</div>

The final section in the input file gives data for the nacelle.

```
! TOWER TOP
nacelle(1,1)    =    0.00    ,    0.000   ,   70.000
nacelle(1,2)    =    120000
nacelle(1,3)    =    27000   ,   1648000 ,   1648000
```

<div style="color:red">x-, y- and z-coordinates
of c.o.g of nacelle

mass moments of inertia of
the nacelle, I_{xx}, I_{yy} and I_{zz}</div>

The first row of this section gives the x-, y- and z-coordinates of the centre of gravity of the nacelle. The second row gives the mass of the turbine (nacelle & rotor). The final row gives the mass moments of inertia of the nacelle, I_{xx}, I_{yy} and I_{zz}.

3.2   Site.dat: Site file

An example of a site file is given below

```
! SITE DATA (LEVELS OF SEA AND MUDLINE)

sitedata(1) =   0.0 ! tide
sitedata(2) = -21.4 ! unscoured mudline
sitedata(3) = -21.4 ! global scour depth
sitedata(4) = -26.4 ! local scour depth
sitedata(5) = -23.4 ! postholing depth
sitedata(6) = -49.0 ! overburden reduction depth

! SOIL

soilname(1) = 'Soil'
soilext  = 'PRO'


! WATER LOADING
w_dens  = 1025
cd      = 0.7
cm      = 2.0
ci      = cm-1
```
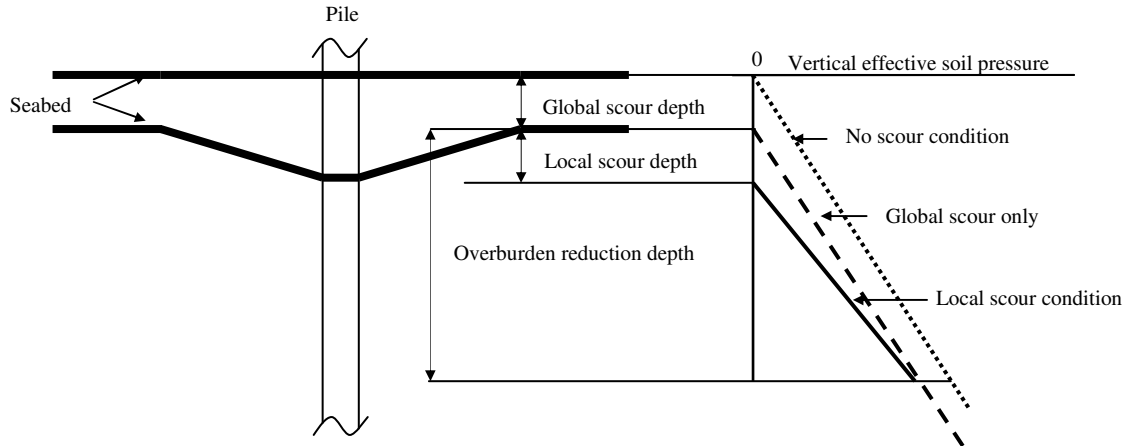
Figure 3: Definition of levels as required in the Site file.

Figure 3 shows the definition of the various levels as required in the site file. The levels are defined relative to a reference plane (usually Mean Sea Level (MSL) or Lower Astronomical Tide (LAT)). *Tide* allows the user to specify the water level with respect to the reference plane. The *unscoured mudline* is the level of the seabed. *Global scour* generally does not occur around a single pile. *Local scour* is likely to occur around a single pile, causing a local reduction of the seabed level around the pile. The absence of material causes a decrease in the soil pressure around the pile. At a certain depth this effect is no longer felt. This depth is the *overburden reduction depth*. In general, a value of 6 times the pile diameter can be assumed.

3.3   Soil.dat: Soil file

An exmple of a soil file is given below:

```
soilprop(1, 1) =    0.00,   sand,  10000,  30.0,  ,   ,   ,   ,          ,
soilprop(1, 2) =   -2.30,   sand,  10000,  35.0,  ,   ,   ,   ,          ,
soilprop(1, 3) =   -6.00,   sand,   9000,  29.0,  ,   ,   ,   ,          ,
soilprop(1, 4) = -14.00,    sand,   9000,  31.0,  ,   ,   ,   ,          ,
soilprop(1, 5) = -16.00,    sand,  10000,  36.0,  ,   ,   ,   ,          ,
soilprop(1, 6) = -25.00,    sand,   9000,  30.0,  ,   ,   ,   ,          ,
soilprop(1, 7) = -27.00,    sand,  10000,  36.0,  ,   ,   ,   ,          ,
soilprop(1, 8) = -36.50,    clay,   9000,       ,  ,   ,   ,   , 175000 ,0.005
soilprop(1, 9) = -38.50,    sand,  10000,  36.0,  ,   ,   ,   ,          ,
soilprop(1,10) = -43.30
```

Depth below Seabed — Soil type — γ' — ϕ — s_u — ε_{50}

γ'	= effective unit weight of soil	[N/m^3]
ϕ	= friction angle	[°]
s_u	= undrained shear strength	[N/m^2]
ε_{50}	= strain at 50% of maximum stress	[-]

The depth of each soil layer is defined relative to the seabed level. Entries are given at each layer transition

Top of first layer   Properties of first layer

soilprop(1, 1) = 0.00   sand,   10000,   30.0,   ,   ,   ,   ,   ,
soilprop(1, 2) = -2.30   sand,   10000,   35.0,   ,   ,   ,   ,   ,

Bottom of first layer and top of second layer

4. Getting started with the macros

Download the macro files from blackboard. Unzip the files and save them in a (new) folder of your choice. Modify the input files to your requirements. Start Ansys. (Ansys can be found by selecting *Programs – Data Analysis & Simulation – Ansys 8.1 – Ansys*). In the Utility menu select *File > change working directory*. Browse to the folder which contains the macros. Before performing a new run select *Utility menu > File > Clear and start new*. Select *do not read file* and click *ok*. Always perform this action before executing the macros. To build the model and perform a modal analysis, select *File > read input from…* from the Utility menu. Select the file *Main.inp*. Ansys proceeds to execute the macros and to build the model and calculate the natural frequencies of the structure. This may take a few moments. When the process is completed a message box containing the text *'solution is done'* appears and the finite element model of the offshore wind turbine is displayed.

5. Performing a modal analysis

During execution of the macros a modal analysis has already been performed. To view the results, select *Main menu > General postprocessor > Results > Results summary*. The first 15 natural frequencies are listed in a new window.

To view the mode shape of the first natural frequency select *Main menu > General postprocessor > Read results > First set*. Subsequently select *Main menu > General postprocessor > Plot results > Deformed shape*. In the '*Plot deformed shape*' window select *def shape only* and click ok.

To view the mode shape of the second natural frequency select *Main menu > General postprocessor > Read results > Next set*. The mode shape can be plotted in the same way as described above. Figure 4 gives the mode shape of the first natural frequency.
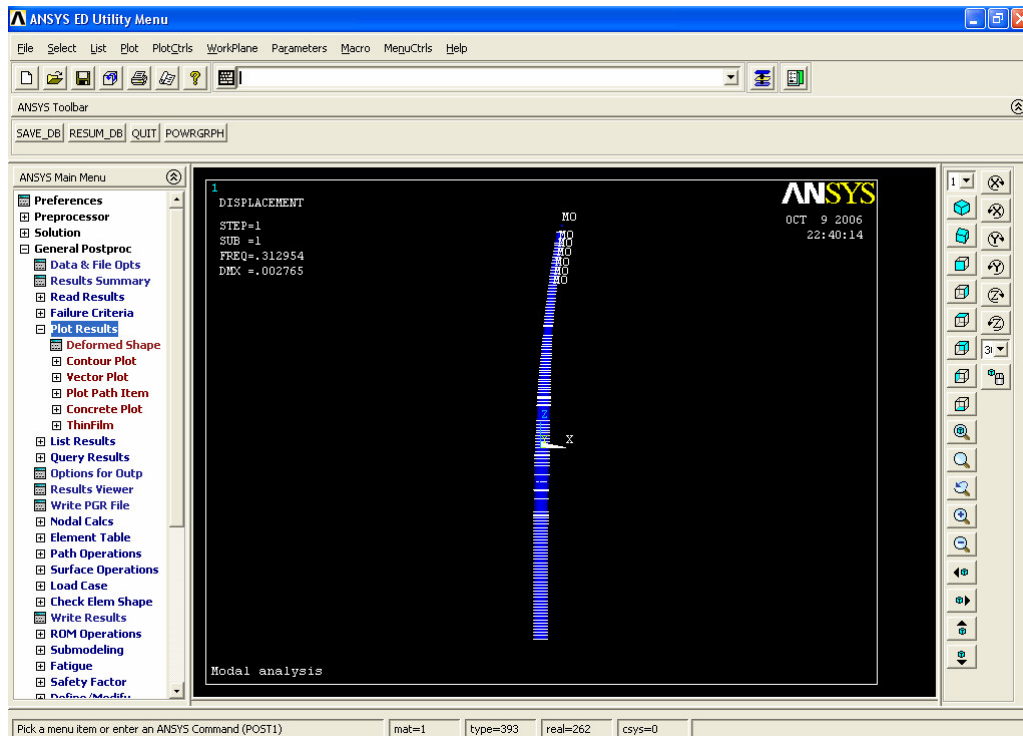

Figure 4: First mode shape

6. Determining the pile penetration depth

6.1 Introduction

To guarantee the lateral stability of the foundation, the pile should enter deep enough into the ground. However, the pile should not be too long as this would result in an uneconomic design. Therefore the penetration depth must be checked. To this end the extreme loads are applied to the model of the structure. Although standards do not give conclusive guidelines on this topic, in general the resulting deflections must satisfy the following conditions:

- The deflection at the mudline is less than a certain value
- The deflection at the pile toe is less than a certain value
- There is a (near) vertical tangent to the deflection curve at some level above the pile toe

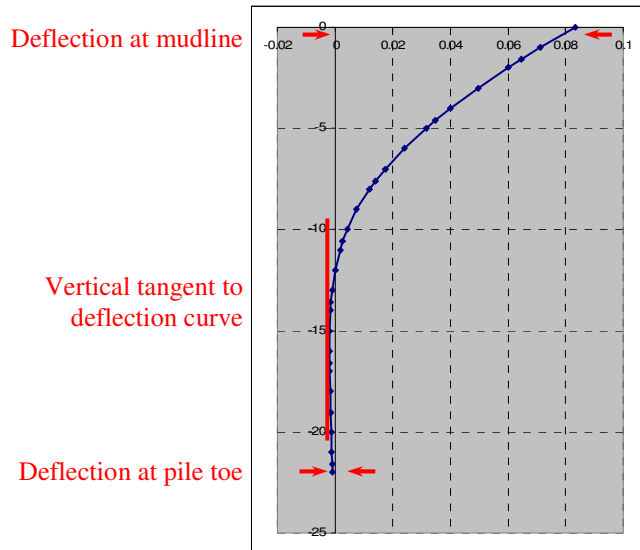This illustrated in figure 5, which gives an example of a deflection curve for a monopile foundation.

Figure 5: Deflection curve of monopile foundation

6.2 Select nodes

The extreme loads must be applied at a certain level. In Ansys concentrated loads are applied on nodes. To find the number of the desired node, the nodes can be listed. This is done by selecting *Utility menu > List > Nodes*. Select *coordinates only* & click *ok*.

The node numbers are listed along with the coordinates. If the loads are to be applied at the mudline the node number at the mudline for the example given in chapter 3 can be found as follows. The mudline is located at MSL -21.40 m. The reference plane is MSL so the required node is at -21.40. The list of nodes shows that the number of the node at the mudline is 90.

```
NODE        X                   Y                   Z
    86    0.00000000000       0.00000000000      -20.0000000000
    87    0.00000000000       0.00000000000      -21.0000000000
    88    0.00000000000       0.00000000000      -21.0000000000
    89    0.00000000000       0.00000000000      -21.0000000000
    90    0.00000000000       0.00000000000      -21.4000000000
    91    0.00000000000       0.00000000000      -22.0000000000
    92    0.00000000000       0.00000000000      -22.0000000000
```

6.3   Apply loads

To apply the loads, select *Main menu > Preprocessor > Define loads > Apply > Structural > Force/Moment > On nodes.* A picking menu appears in the bottom left corner of the screen. Enter the number of the node to which the force is to be applied and click *ok*. The '*Apply F/M on nodes*' window appears. Select the direction of force/moment and enter a value. Figures 6a and 6b show the node picking menu and the '*Apply F/M on nodes*' window respectively. Be aware that F_x means a force in the *x*-direction and M_x implies a bending moment around the *x*-axis. F_x and M_x do not act in the same plane!

To verify that the loads are entered correctly, the loads can be listed by selecting *Utility menu > List > Loads > Forces > On all nodes*. Select *coordinates only* & click *ok*.
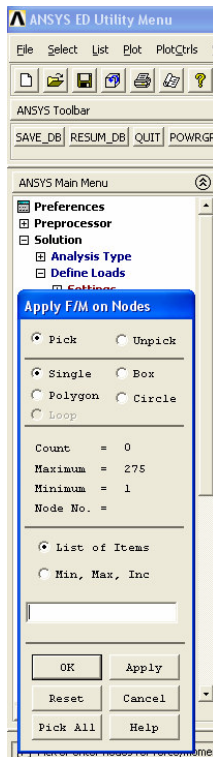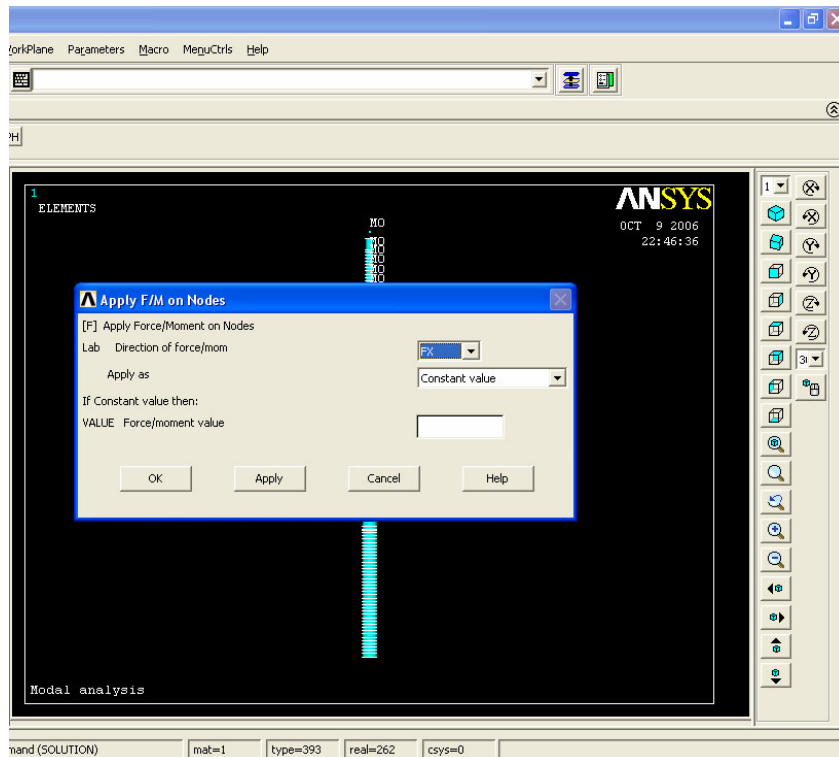

Figure 6a: *Node picking menu*   **Figure 6b:** *'Apply F/M on Nodes' Window*

6.4   Solution

Before solving the current problem the analysis type must be set to static. Select *Main menu > Solution > Analysis type > New analysis.* Select *Static* & click *ok*. See figure 7. To solve the

problem select *Main menu > Solution > Solve > Current load step > ok.* After a few moments a message box appears containing the message '*Solution is done*'.

6.5 Results

Results can be listed by selecting *Main menu > General postprocessor > List results > Nodal solution.* The '*List nodal solution window*' appears. This is depicted in figure 7.
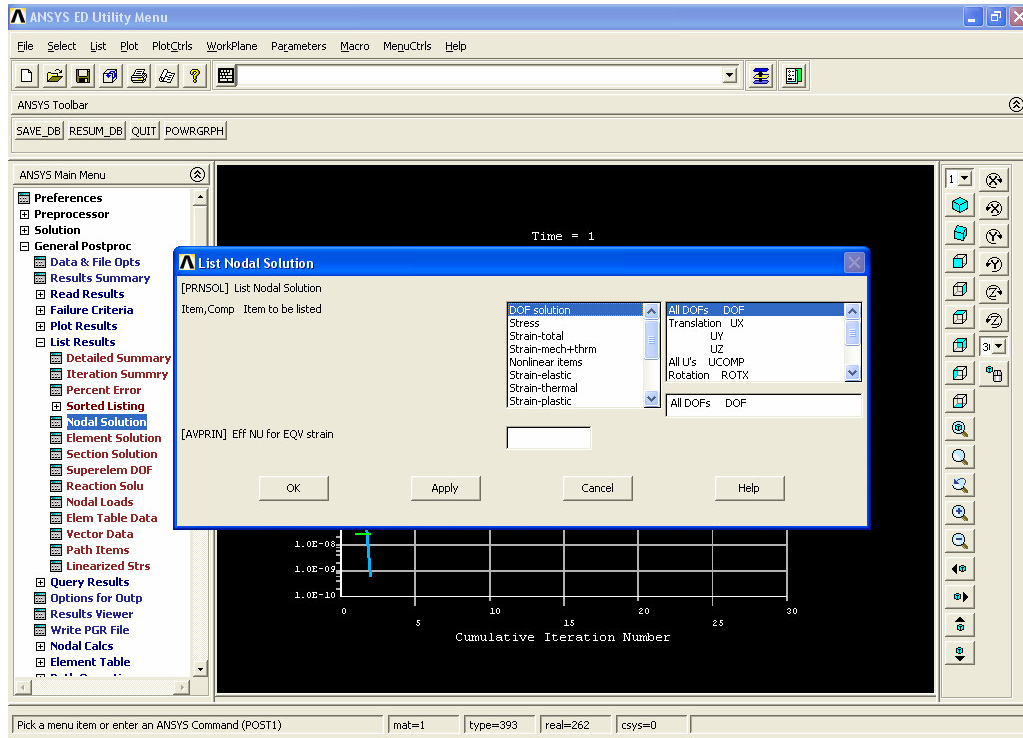

Figure 7: *The 'List nodal solution' window*

Horizontal deflections for the nodes around the mudline and near the pile toe are listed below as well as the rotations near the vertical tangent to the deflection curve. The vertical tangent occurs where the rotations change sign as is shown in the third column.

NODE	UX	NODE	UX	NODE	ROTY
79	0.22081E-02	260	-0.15506E-05	194	0.13469E-05
80	0.22200E-02	261	0.0000	195	0.0000
81	0.23033E-02	262	0.0000	196	0.0000
82	0.23476E-02	263	-0.76442E-06	197	0.87775E-06
84	0.44057E-03	264	0.0000	198	0.0000
85	0.38347E-03	265	0.0000	199	0.0000
86	0.29212E-03	266	0.0000	200	0.0000
87	0.26928E-03	268	-0.11276E-08	202	0.48671E-06
88	0.0000	269	0.0000	203	0.0000
89	0.0000	270	0.0000	204	0.0000
90	0.26015E-03	271	0.0000	205	0.0000
91	0.24453E-03	273	0.76128E-06	207	0.16000E-06
92	0.0000	274	0.0000	208	0.0000
93	0.0000	275	0.0000	209	0.0000
94	0.21867E-0	276	0.0000	210	0.0000
95	0.0000	278	0.0000	212	-0.67568E-07
96	0.0000	280	0.0000	213	0.0000

The deformation of the structure can be visualized by selecting *Main menu > General postprocessor > Plot results > Deformed shape.* The resulting plot will be similar to figure 8.
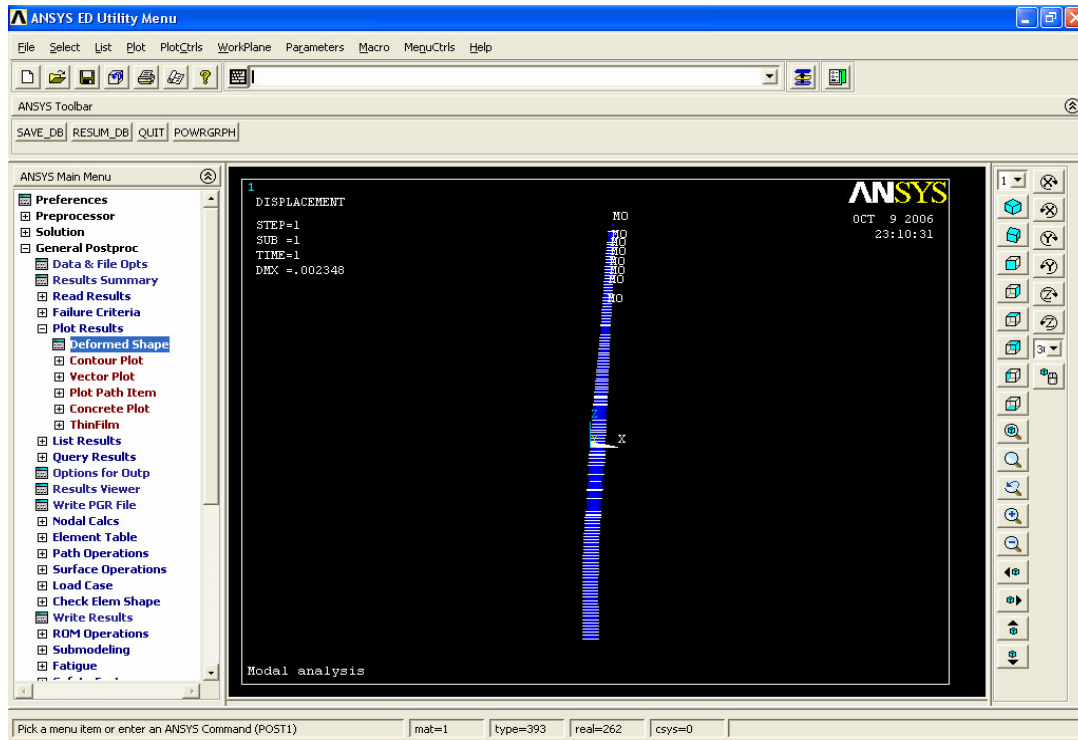

Figure 8: *Deflection of structure with loads applied at seabed.*

6.6 Iteration

If the deflections satisfy the criteria given in section 6.1, the pile should be shortened in the Struct file and the macros should executed again. (Remember to select *Utility menu > File > Clear and start new*) Repeat the process described above until the criteria are not satisfied. The penetration depth is the smallest depth for which the criteria are met.

7. Determining the translational and rotational ground springs

7.1 Introduction

For later use in the turbine simulation program Bladed the translational and rotational ground springs must be determined. In Ansys the soil stiffness is modelled as shown on the left side of figure 9. In Bladed the springs are modelled as shown on the right of this figure.
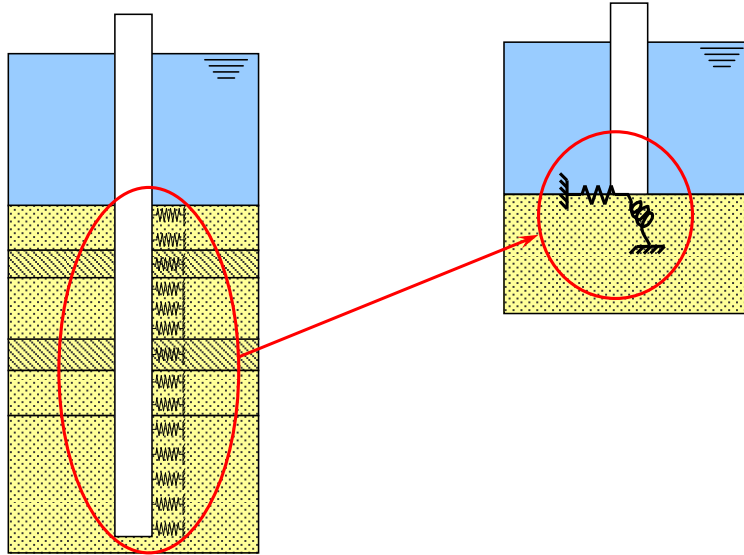

Figure 9: *From soil springs in Ansys to soil springs in Bladed.*

7.2 Translational ground spring

The translational ground spring for use in Bladed can be found by applying a horizontal force *F* at the mudline and determining the resulting deflection *u*. The translational ground spring k_u can be found with equation 1.

$$k_u = \frac{F}{u} \tag{1}$$

How to apply the load and how to obtain the results has already been explained in chapter 3. (Make sure the analysis type is set to *static*)

7.3 Translational ground spring

The translational ground spring for use in Bladed can be found by applying a bending moment *M* at the mudline and determining the resulting rotation θ. The rotational ground spring k_θ can be found with equation 2.

$$k_\theta = \frac{M}{\theta} \tag{2}$$