Abstract: This paper presents the calculation of a bridge shallow foundation using the finite element method together with a nonlinear procedure. In order to design a shallow foundation for a bridge, it is necessary to establish, based on soil type and its characteristics, the effective maximum pressure on the soil below the foundation base. The problem can become difficult to solve when negative pressure on the soil are expected. In these cases the “effective area” concept should be used, but even this method is difficult to be applied when the normal force acting on the foundation is eccentric according to both axes in plan. For this reason, in this paper is presented a finite element model which can lead through a nonlinear calculation to correct results.

Key-Words: bridge, shallow foundation, soil type, foundation, normal force, soil pressure

1 Introduction

The choice of the foundation system for bridges substructure is influenced by the soil type and characteristics. It is well known the fact that for a new bridge, the costs for foundations works rise to 30-40% from the total cost. In these conditions, according to the soil type and characteristics known from geotechnical drillings, two solutions can be considered: the execution of a shallow foundation with lower costs or a deep foundation on piles which rises the costs level.

This paper presents, from the calculation point of view, the case of shallow foundations for bridges. In order to validate the adopted solution for shallow foundations, it is necessary to establish the maximum value of the effective pressure on the soil under the foundation mat. The pressure value can be established using traditional analytical methods based on the existing standards and norms. Although, if tension stresses caused by the high values of forces acting on the foundation mat occur, the calculation should be based on the “effective zone” concept.

Moreover, if the eccentricity of the vertical force in two in plan orthogonal directions is significant, than the real distribution of the pressure on the soil is difficult to establish using the analytical methods mentioned above.

In this paper, a simple direct approach based on the finite element method and nonlinear analysis for establishing the distribution of pressures under the mat of a shallow foundation of a bridge on Transylvania motorway is presented.

2 Description of the bridge structure

The structure analyzed in this paper is a viaduct designed on the Transylvania motorway, section Târgu Mureş-Cluj-Napoca, between km 18+502.42 and km 18+659.72.

In plan view the bridge (Fig.1) is placed on a curve having a radius of 1600 m, in elevation has a longitudinal slope of about 5% and was designed with four spans: lateral spans of 38.55 m and intermediate spans of 40.00 m. The substructure consists in two massive abutments A1 and A2 and three piers P1, P2 and P3 having a double box cross section (Fig.2), all made in reinforced concrete of class C35/45. The height of the piers elevation varies in the range 22 to 40 m. The superstructure for one way was designed in the solution with four juxtaposed precast “U” beams (Fig.2). Over the beams a reinforced concrete slab having a thickness of 25 cm was foreseen. The depth of the precast concrete girder is 2.20 m and the width of the carriageway is 12.00 m. For the bearing of the superstructure on the substructure elements elastomeric bearing with a height of 63 mm were used.
Following the geotechnical drillings on the site it was found that starting with the depth of 3m below the ground level a hard limestone is present, so that the solution with shallow foundations appear to be more effective for the design. The foundation mat has the following dimensions: 8.00m in longitudinal direction, 10.00m in transverse direction and is 2.00m thick.

Fig. 1 General layout of the bridge

Fig. 2 Cross section of the bridge
3 Analytical method for establishing the maximum values of the pressure under the foundation mat

Usually, for establishing the maximum values of the pressure under a shallow foundation mat Navier’s formula has to be used, considering the total vertical force and bending moments in two orthogonal directions in plan, acting in the center point of the foundation mat. Although, there are cases when the eccentricity of the vertical force in both directions is high enough and the point where the load acts is placed outside the cross section central core. As a consequence, the values of the bending moments lead to tensions in soil on some parts under the foundation mat. In this case, the calculation should use the “effective zone” concept considering that the vertical force is totally taken by the soil in the compression zone.

In [1],[3] and [4] is stated that starting form the Navier-Bernoulli hypothesis the normal stresses on a cross section have a linear distribution on the “effective zone” under the foundation, with zero value in the neutral axis and a maximum value on the edge (Fig. 3a). For an arbitrary cross section, the value of the stresses can be determined writing the equilibrium equations between the external vertical force $N$ and the resultant force produced by the volume of the normal stresses (Fig. 3b).

\[
\int \sigma \ dA = N
\]

(1)

and with respect to the neutral axis the following equation exists:

\[
N \cdot d = \int \sigma \cdot ydA
\]

(2)

The stresses are proportional with the distance to the neutral axis and considering the figure 3a) it results:

\[
\sigma = \sigma_{\max} \cdot \frac{y}{y_{\max}}
\]

(3)

By introducing (3) in (1) it results the formula of the maximum stresses as a function of the vertical force value and geometrical characteristics of the section as follows:

\[
\sigma_{\max} = \frac{N \cdot y_{\max}}{S_{za}} = \frac{N \cdot d \cdot y_{\max}}{I_{za}}
\]

(4)

with

\[
I_{za} = S_{za} \cdot (d - e)
\]

(5)

In the above equations, $S_{za}$ and $I_{za}$ are the static moment of area and the first moment of inertia of the “effective zone” with respect to the neutral axis.

If the cross section of the foundation mat has rectangular form (Fig.3b) (the most common situation), then there are several possible positions...
of the vertical force $N$ with respect to the limits of the central core of the section:

- If the point $A$ is placed on the symmetry axis, at a distance $c$ with respect to the closest edge of the section, then the value of the stress can be established using the relationship below:

$$\sigma_{\text{max}} = \frac{2N}{3Bc} \quad (6)$$

- When the point $A$ of application of force $N$ is placed inside the central core of the cross section, having the coordinates $x_0$ and $y_0$ with respect to the coordinates system $xOy$ (Fig.4), the maximum stress in the corner of the foundation mat where the force $N$ acts can be calculated on the basis of the formula:

$$\sigma_{\text{max}} = \mu \frac{N}{A} \quad (7)$$

where $A$ is the cross section area of the foundation mat and $\mu$ is a coefficient as a function of the ratios $\frac{x_0}{L} \cdot \frac{y_0}{B}$. The relationship (7) still remain acceptable even when the point of application of the force is outside the central core of the cross section.

4 Numerical models and performed analyses

The analyses were performed by the aid of finite elements method. In order to validate the chosen solution for the foundations, the analysis were performed in two steps:

- In the first step, using the model for the whole structure, a linear static analysis was conducted, considering all loads acting during service life of the bridge and considering that the piers are fully fixed at the base;

- In the second step, using the internal forces calculated in the previous step, a nonlinear static analysis was performed. Only the foundation mat was modeled and loaded with the set of forces acting on the cross section between the elevation of the pier and the foundation mat.

For the analysis performed in the first step two types of models were considered based on the type of loads and the way they act on the foundation: a model corresponding to the slow actions (which are developing in time similar to the shrinkage of the concrete) and a model for short term actions.

The model for the whole structure (Fig.5) was built so that it respects the static scheme of the structure: simply supported beams in the erection stage of the bridge and continuous girders after pouring the concrete slab above them.

The deck was modeled using frame elements with two joints, having six degree of freedom at each joint (three translations and three rotations respectively). The frame elements were placed in the centroid of the whole bridge superstructure and they include the geometrical characteristics of the complete cross section of the bridge, including the four precast prestressed “U” beams and the concrete slab above them. Above the piers, the frames modeling the deck were interrupted and in order to model the effect of the continuity of the concrete slab a system of “dummy beams” was built. The beams connect the superstructure with the substructure of the bridge, but ensure also the continuity of the concrete slab in each section above the piers, where a gap between girders ends exists. (Fig.6).
The finite elements modeling the slab are similar to those used for the deck, but disposed eccentrically with respect to the neutral axis of the superstructure in the span, the value of the eccentricity representing the vertical distance between the centroid of the slab and of the superstructure. This is the length of the finite elements connecting these two parts of the model. The supporting saddles which ensure the bearing of the bridge superstructure were modeled in the same way, using two joint straight frame finite elements and the connection with the piers elevation was made at a joint placed in the piers longitudinal axes. Between the superstructure and piers supporting saddles "dummy beams" were used to ensure the connection.

The finite elements connecting the superstructure in the span and the concrete slab above the piers, the superstructure and the substructure, but also those which are modeling the supporting saddles were considered with a high stiffness. The stiffness characteristics of the finite elements used for the concrete slab are based on the real geometry and dimensions of the slab.

Two joints finite elements were used also for modeling the abutments and piers.

The elastomeric bearing devices were introduced into the numerical model by the aid of two joint link elements (Fig.6). For these, the values of axial, bending and shear stiffness were calculated based on the physico mechanical properties of the material (longitudinal and transverse moduli of elasticity). The shear stiffness was introduced into the analyses separately, considering the appropriate value according to the duration of the loading acting on the structure: long or short time actions respectively. The behaviour of the link elements modeling the bearing was assumed to be linear elastic.

Following the performed analyses, the values of the internal forces in all structural elements of the bridge were obtained.

In the second step, in order to evaluate the maximum values of the pressure on the soil under the foundation mat, two separate finite element models were built. The mat was considered very stiff and soil-structure interaction was considered in two ways: with elastic springs and with nonlinear link (gap) finite elements.

The link element described in [5] can be used in two cases: to connect two joints and this was the case for the bearing devices, but also for connecting the joints of the mat with the soil surrounding it and this one was the cases for considering the soil-structure interaction. Each link consists in a set of six springs, each corresponding to one degree of freedom, three translations and three rotations.

The six internal deformations of the link finite element (Fig. 7) are determined using the relative displacements of the element joints i and j respectively.

For a two joint link finite element, the internal deformations can be described using the following formulas:

- Axial deformation
  \[ du_1 = u_{1j} - u_{1i} \]  
  \[ du_2 = u_{2j} - u_{2i} - d j 2 \cdot r_{3j} - (L - d j 2) \cdot r_{3i} \]
- Shear deformation in plane 1-2
  \[ du_3 = u_{3j} - u_{3i} + d j 3 \cdot r_{1j} + (L - d j 3) \cdot r_{1i} \]
- Deformation from torsion:
  \[ dr_1 = r_{1j} - r_{1i} \]
- Bending deformation in plane 1-3:
  \[ dr_2 = r_{2i} - r_{2j} \]
– Bending deformation in plane 1-2:

\[ dr_j = r_{3j} - r_{3i} \]  

(14)

In the above formulas \( dj \) and \( dj \) are the distances from the joint \( j \) to the section along the element where the shear deformation occur and \( L \) is the length of the link finite element.

Using the deformations described through the formulas (9)-(14), the relations force-deformation for all degrees of freedom can be written under the form bellow:

\[
\begin{bmatrix}
    f_{u1} \\
    f_{u2} \\
    f_{u3} \\
    f_{r1} \\
    f_{r2} \\
    f_{r3}
\end{bmatrix} =
\begin{bmatrix}
    k_{u1} & 0 & 0 & 0 & 0 & 0 \\
    0 & k_{u2} & 0 & 0 & 0 & 0 \\
    0 & 0 & k_{u3} & 0 & 0 & 0 \\
    0 & 0 & 0 & k_{r1} & 0 & 0 \\
    0 & 0 & 0 & 0 & k_{r2} & 0 \\
    0 & 0 & 0 & 0 & 0 & k_{r3}
\end{bmatrix}
\begin{bmatrix}
    du1 \\
    du2 \\
    du3 \\
    dr1 \\
    dr2 \\
    dr3
\end{bmatrix}
\]  

(15)

where \( k_{u1}, k_{u2}, k_{u3}, k_{r1}, k_{r2}, k_{r3} \) are the values of the stiffness for each degree of freedom.

The element internal forces are to be determined as follows:

\[
\begin{bmatrix}
    N \\
    T_2 \\
    T_3 \\
    M_1 \\
    M_2 \\
    M_3
\end{bmatrix} =
\begin{bmatrix}
    k_{u1} & 0 & 0 & 0 & 0 & 0 \\
    0 & k_{u2} & 0 & 0 & 0 & -dj2 \cdot k_{s2} \\
    0 & 0 & k_{u3} & 0 & -dj3 \cdot k_{s3} & 0 \\
    0 & 0 & 0 & k_{r1} & 0 & 0 \\
    0 & 0 & 0 & k_{r2} + dj3 \cdot k_{s3} & 0 & 0 \\
    0 & 0 & 0 & 0 & k_{r3} + dj2 \cdot k_{s2} & 0
\end{bmatrix}
\begin{bmatrix}
    \nu_1 \\
    \nu_2 \\
    \nu_3 \\
    \nu_4 \\
    \nu_5 \\
    \nu_6
\end{bmatrix}
\]  

(16)

where \( N, T_2, T_3, M_1, M_2, M_3 \) are element internal forces.

For the link-gap finite element (Fig.8) all internal deformations are independent, the relation force-displacement being described through the equations:

\[
f = \begin{cases} 
  k \cdot (d + \Delta), & \text{if } (d + \Delta) < 0 \\
  0, & \text{if } (d + \Delta) > 0 
\end{cases}
\]  

(17)

where \( d \) is the element deformation.

The foundation mat was modeled using only joints with rigid body constraints. (Fig.9). In these joints elastic spring elements were introduced in the first finite element model and link (gap) finite elements in the second model. The stiffness in vertical direction of springs and link elements was calculated on the basis of proportionality coefficient “m” of the soil under the foundation mat and considering the appropriate area surrounding each spring or link element.

The vertical forces obtained from the static analysis of the whole structure were placed in the joint representing the center of the foundation mat (Fig.9). Following the linear and nonlinear analyses using the finite element models of the foundation mat, the values of the reactions at each joint were obtained and subsequently these were converted in pressure values.

![Fig.8 Scheme of a link (gap) finite element [5]](image)

![Fig.9 Finite element model of the foundation mat](image)

In order to validate the obtained results using these finite element models and based on the theoretical aspects presented at point 3 of this paper, a computer program was written in Borland Pascal. This program gives the maximum value of the pressure on the soil under the foundation mat for any arbitrary position of the vertical force acting on the foundation. The values of the pressure obtained using the finite element models and the computer program are summarized in Table 1.

| Table 1 Values of the pressure on the soil under the foundation mat |
|------------------|-------------|
| Finite element model | Maximum pressure, [kPa] |
| Using springs      | 550.00      |
| Using links (gaps) | 568.50      |
| Computer program   | 577.90      |

The distribution of the pressure on the soil under the foundation mat following performed analyses is shown in figure 10: case a) using spring finite elements, case b) using link (gap) finite elements.
5 Conclusions

As a result of performed analyses it can be observed that the values of the pressure on the soil under the foundation mat are slightly changing (in Table 1 the differences are around 5%) when using the “effective zone” concept and link (gap) elements together with performing a nonlinear analysis. This difference was obtained for common values of forces applied on the foundation, leading to a small to moderate value of the eccentricity. But, the distribution of the pressure changes as presented in figure 10 and it can change significantly for big eccentricity of vertical force, which can be the case for bridge piers foundations that are not very well designed concerning the position of the abutment elevation with respect to the center point of the foundation mat. The use of spring finite elements for modeling soil-structure interaction of shallow foundations should be done with caution in order to not overestimate the bearing capacity of the soil. In the same time, the use of link (gap) finite elements with appropriate values of stiffness together with a full nonlinear analysis is much more realistic an lead to correct results.

References: