Authors: Doc. Ing. Radim Čajka, CSc; Ing. Ondřej Víta, VŠB – Technical University of Ostrava

ELEMENT FOR MODELLING OF INTERACTION BETWEEN FOUNDATIONS AND BEDROCK IN ANSYS SOFTWARE

Summary

Ansys provides standard solutions to contact tasks. It is a high-performing computing tool that offers several modifications, making it possible to create a new element and adjust the calculation to individual requirements. When dealing with specific problems relating to the contact between the foundation and the foundation soil, it is possible to create an element that will take into account the behaviour of a soil layer under the foundation exposed to shear. The shearing load can be caused by undermining, concrete creep, or concrete shrinkage.

Element Geometry

When modelling the interaction between the foundation and the bedrock, a suitable element is that with the geometry depicted in Figure 1. Such an element is required to transfer tensions in the axial direction, this means in the direction normal to friction surfaces, and shearing tensions in the direction normal to the axis of the element, this means in the direction of the friction surfaces.

The element has got two degrees of freedom in each of the two nodes, namely the shifts in x and y directions.

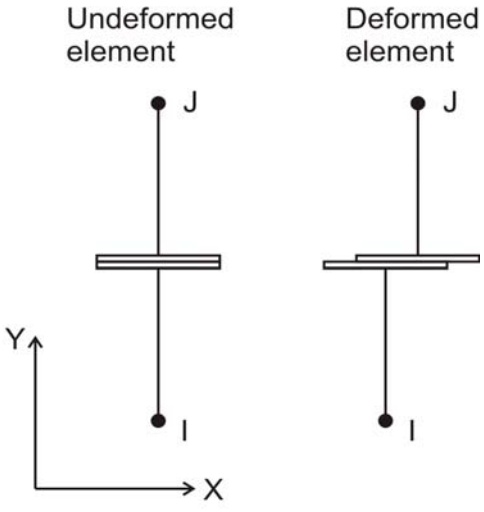

Fig 1. – Element geometry

Real Constants and Material Properties

Foundations with a variable width can be solved only if one of the element parameters is the width of the foundation. An example of such a foundation and the model are shown in Fig. 2.

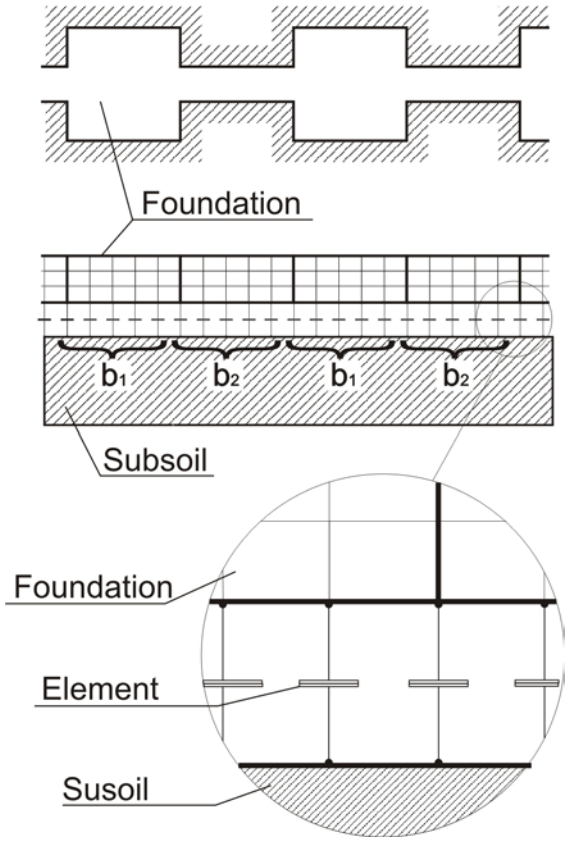

Fig 2. – Model of foundation

Other parameters must include φ (internal fiction angle) and c (cohesion). Those parameters are necessary for the calculation of the limit shearing friction. It is also essential to know E_{oed}

This outcome has been achieved with the financial support of the Ministry of Education, Youth and Sports of the Czech Republic, project No. 1M0579

(oedometric modulus of the foundation soil) as it is used for the calculation of shearing tensions.

Definition of Element Behaviour

Because the element will be loaded with the forced deformation only (with the shift of one node only), it is necessary to define the behaviour of the element in dependence on the shift of the nodes, shear strength of the element, and axial force transferred by the element. In contrast to the calculation pursuant to the standard ČSN 73 0039 the calculation does not take into account ε_{eig} (deformation of concrete).

So that the calculation could be in line with real conditions, it is essential to define two various states of the element and define a stiffness matrix based on those two states. The first state is the situation when the limit shearing stress has not been achieved yet, and the shearing stress goes up in non-linearly in a directly proportional manner to the deformation. The second state is when the limit shearing stress has been achieved, the shearing stress will not increase any more, and the element transfers the constant shearing stress, independently of the magnitude of the deformation. The dependence of the shearing force transferred by the element to the shift of the nodes is depicted in Fig. 3.

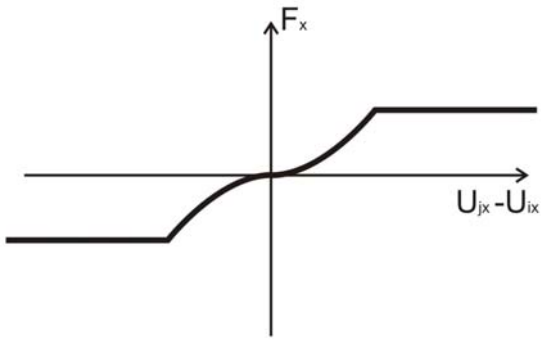

Fig 3. - Dependence of friction force on shift of nodes

Fortran77 and C++ are programming languages used in Ansys. Source codes need to be compiled and integrated into Ansys.

In the uec100 file, the following parameters need to be defined:

- 2-D and 3D geometries
- degrees of freedom
- symmetry of matrixes
- number of nodes
- volume load
- surface load
- number of real constants
- number of variables to be saved
- number of matrix lines for the element
- whether the behaviour of the element is linear or non-linear

The uel100 file defines the method used for the calculation of element matrixes (for instance, the stiffness matrix), the element load vector, and the output quantities of the element.

The uep100 file provides the output for the elements used. It is recalled by the post-processor.

The usertr file makes it possible to access node transformations.

The userac file describes the work with data.

Legislation and standards

- ČSN 73 0039 Designing Buildings on Undermined Territories, Publishing House for Standards, Prague 1990

Bibliography

- Bradáč, J.: Effects of Undermining and Protection of Structures (in Czech), EXPERT – technical publishing house Ostrava, Ostrava, 1996
- ANSYS Release 8.1 Documentation Preview
- ANSYS APDL Programmer's Guide, http://www.ansys.com/Documentation/Manuals/872
- The UIDL Programmer's Guide, http://www.ansys.com/Documentation/Manuals/872
- Pěnčík, J.: Physical Non-Linear Analysis of Concrete Plane Framework –dissertation propositions (in Czech), ISBN 80-214-1898-2, ISSN 1213-4198
- Ansys Theory Guide 5.7.1 , spc.vsb.cz
- Lenert, J.: Introduction to the Finite Element Method (in Czech), VŠB – Technical University Ostrava, Ostrava, 1999, ISBN 80– 078–686 - 8

This outcome has been achieved with the financial support of the Ministry of Education, Youth and Sports of the Czech Republic, project No. 1M0579