Optimised design of foundations for wind towers

L. Lacoma, J. Rodríguez, F. Martínez and J. Martí

PRINCIPIA Ingenieros Consultores

***Abstract**: Environmental, political, and other considerations are leading to the installation of growing numbers of wind turbines for electricity generation. The present paper concentrates on onshore installations, in which the turbine is placed at the end of a tower founded on the ground. A serious problem with the foundation or its connection with the tower entails a global failure.*

A worrying percentage of the towers built before 2010 is ill designed, often with obvious manifestations of the design problem. On the other hand the new designs attempt to optimise material quantities and costs while providing an adequate behaviour under the demands imposed.

Principia has gathered considerable experience in both the identification and remediation of the problems experienced by older foundations and the optimisation of new designs. The problem is particularly suitable for numerical simulations with Abaqus and the paper provides the necessary concepts and methodological approach for a reliable and satisfactory solution. More specifically, the concrete geometry is meshed with solid elements. Surface elements including rebar layers are embedded. Other parts of the structure as the steel cylinder, welded studs, anchorage bars, etc. are also included. The concrete behaviour is described with the Abaqus concrete damaged plasticity model. Abaqus/Explicit is used because of the strong non-linearities, mainly caused by concrete cracking and crushing. The methodology allows assessing the suitability of the design for ultimate, serviceability and fatigue limit states.

***Keywords**: Concrete, Foundations, Rebars, Wind Turbines*

1. Introduction

Environmental, political, and other considerations are leading to the installation of growing numbers of wind turbines for electricity generation. The present paper concentrates on onshore installations, in which the turbine is placed at the end of a tower founded on the ground.

Wind turbine towers are isostatic structures. They have no redundancies, in the sense that a serious problem with the foundation or its connection with the tower entails a global failure of the tower. This endows the design of the foundation and its connection to the tower with an especial responsibility.

Because of their participation in the overall cost, the mechanical and electrical components tend to receive considerably more attention than the civil infrastructure. This has many consequences, one of them being that a worrying percentage of the towers built before 2010 is ill designed, often with obvious manifestations of the design problem.

On the other hand, as the market matures, new designs need to become progressively more competitive and attempt to optimise material quantities and costs while providing an adequate behaviour under the demands imposed.

Whether because of the need to understand and overcome the shortcomings of some existing structures or because of the perceived requirement to optimise material quantities, it has become necessary to be able to conduct realistic analyses of the behaviour of those foundations in relation with the various limit states, including ultimate, serviceability and fatigue limit states.

Principia has gathered considerable experience in both the identification and remediation of the problems experienced by older foundations and in the optimisation of the new designs. Some of this experience is conveyed in the present paper, using an example of a defective foundation in order to transmit the information. As will be seen the types of problems posed are particularly well suited for numerical simulations with Abaqus.

2. Methodology

In order to simulate the problems with the necessary accuracy, the concrete and the steel must be represented and the basic characteristics of their behaviours and interaction must be adequately modelled.

In the methodology adopted by Principia, the concrete geometry is meshed with solid elements. Surface elements are embedded in order to represent the reinforcement layers. Other parts of the structure as the steel cylinder, welded studs, anchorage bars, etc. must also be included.

From the viewpoint of the mechanical behaviour concrete is characterised with the Abaqus concrete damaged plasticity model, while traditional elastoplastic descriptions are used for the various steel elements.

The problem is best analysed using the Abaqus/Explicit solver. This is motivated by the strong nonlinearities of the problem, primarily caused by concrete cracking and crushing.

Additional details are given in the subsequent sections, taking a specific case as an example to clarify the application of the methodology described. The case is one involving defective foundations of existing wind towers, which had to be investigated and for which remedial measures had to be developed.

2.1 Geometry and mesh

The geometry of the problem being studied is shown in Fig. 1. It includes the footing with its reinforcement, the lower part of the shaft with the shell and flange designed to provide the connection with the footing.

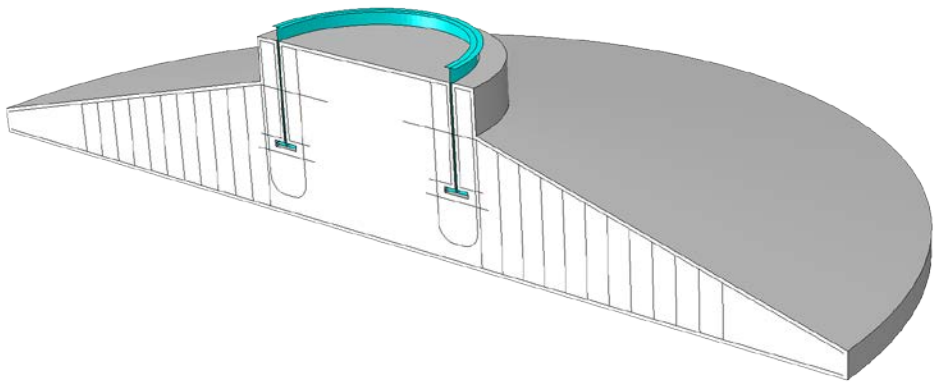

Figure 1. Geometry of the foundation.

As mentioned earlier solid elements C3D8R were used to represent the concrete. The surface elements employed for modelling the reinforcement are SFM3D4R elements; their distribution can be seen in Fig. 2.

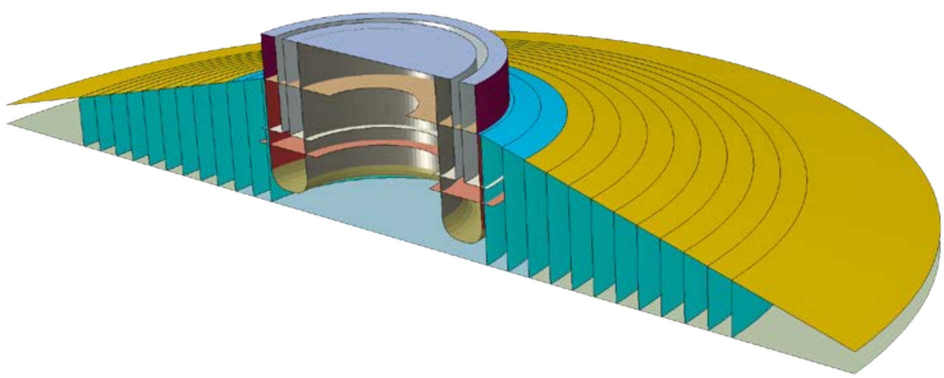

Figure 2. Surfaces with rebars embedded in the concrete.

The steel shell and flanges were represented by means of shell elements S4R. A general view of the global mesh is presented in Fig. 3. The total number of elements in the mesh is 150,000, with 155,000 nodes and about 475,000 degrees of freedom.

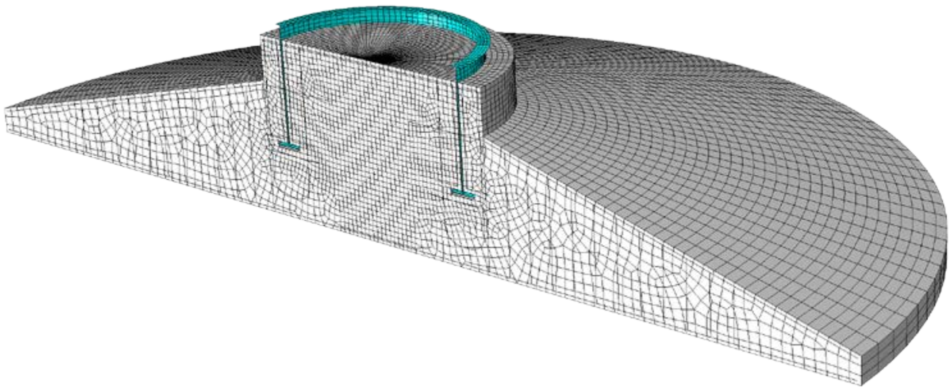

Figure 3. Finite element mesh.

Although not dealt with in the present paper, it should be noted that other elements such as post-tensioned bolts, often incorporated as a remediation strategy, can easily be simulated as well. In that case, they would be modelled using beam elements B31 or truss elements T3D2. As an example Fig. 4 shows a geometry in which those existed.

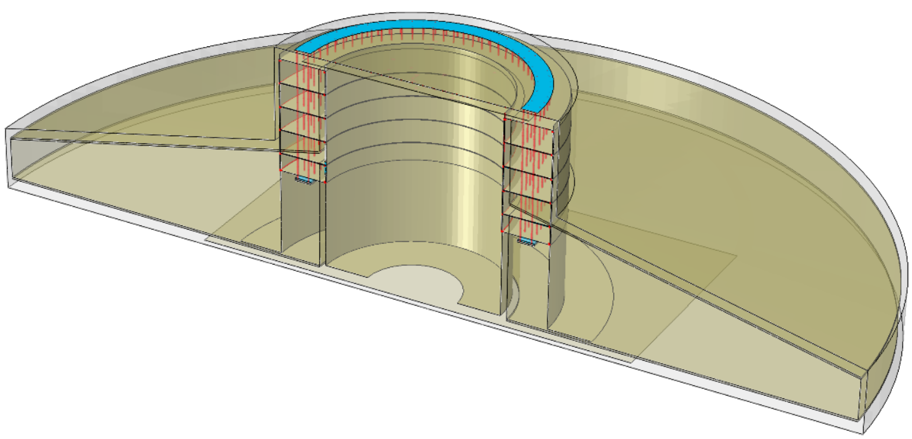

Figure 4. Geometry of a foundation with post-tensioned bolts.

2.2 Materials

The concrete damaged plasticity model in Abaqus provides a general capability for the analysis of concrete structures under monotonic, cyclic, and/or dynamic loading. It includes a scalar (isotropic) damage model with tensile cracking and compressive crushing modes. The model accounts for the stiffness degradation mechanisms associated with the irreversible damage that occurs during the fracturing process. Some of the more salient features of the damaged plasticity model can be conveyed with a few figures, as attempted below.

Fig. 5 shows a representative response of the model under monotonically increasing uniaxial strains according to [1]; the upper graph in the figure presents the expected brittleness of the tensile response, while the lower graph describes the behaviour under uniaxial compression.

The post-failure response is controlled by a fracture energy. The reference value of the fracture energy is given by the Model Code 2010 as $73 \cdot f_{cm}^{0.18}$, equal to 140 J/m^2 for the actual concrete.

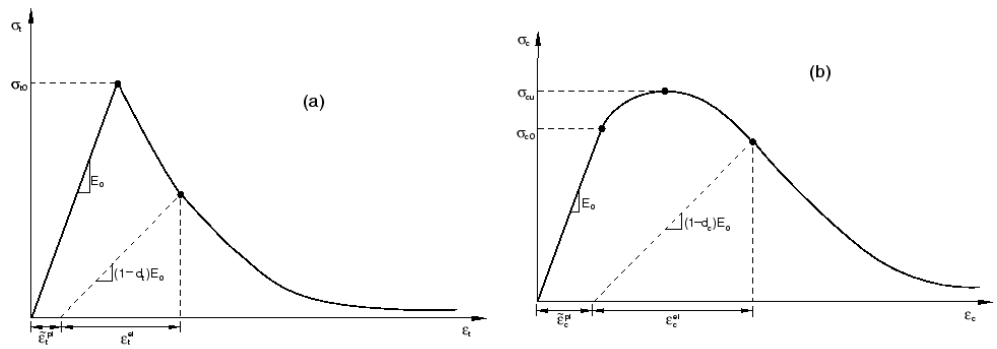

Figure 5. Uniaxial loading in the CDP model in (a) tension and (b) compression

Due account is taken of the effects of multiaxial compression, as shown by the yield surface presented in Fig. 6, which corresponds to a plane stress situation.

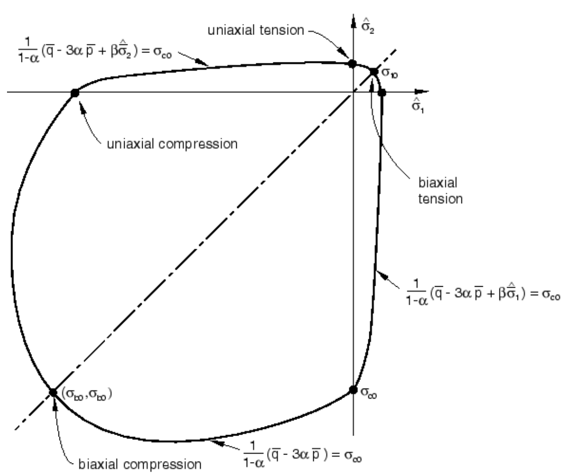

Figure 6. Yield surface in plane stress.

Since cyclic or reversal effects are not considered important in the simulations, no damage (elastic stiffness degradation caused by the mechanical strain history) is defined in the model.

The concrete used in the footing and the pedestal is C30/37 concrete, and the mechanical properties, reduced by a factor of 1.5, are given in the Table 1.

Table 1. Material properties of concrete

Property	Concrete C30/37
Young´s modulus (GPa)	28.4
Poisson´s ratio (-)	0.3
Density (kg/m^3)	2400
Compressive strength (MPa)	20
Crushing strain (-)	0.35%
Dilation angle (º)	30
Tensile strength (MPa)	1.33
Fracture energy (J/m^2)	100

The steel elements have been characterised by means of an elastoplastic constitutive model based on the von Mises yield criterion with an associative flow rule. For rebars, a BSt 500 S steel is used. The material properties assigned, already reduced by 15% for the calculations, are given in Table 2.

Table 2. Material properties of reinforcing steel

Property	Steel BSt 500 S
Young´s modulus (GPa)	200
Poisson´s ratio (-)	0.3
Density (kg/m^3)	7850
Yield stress (MPa)	435
Ultimate elongation (-)	2.25%

Besides, the steel in the flanges and the cylindrical shell is described as S355 with the mechanical properties in Table 3.

Table 3. Material properties of steel S355

Property	Steel S355
Young´s modulus (GPa)	200
Poisson´s ratio (-)	0.3
Density (kg/m^3)	7850
Yield stress (MPa)	309

The soil is assumed to have a representative ballast modulus of 10 MN/m^3 for the actual foundation area and a friction coefficient of 1.0. This is implemented with a soft contact (implemented with a contact pair) between the bottom face of the foundation and a rigid body fixed horizontal plane. The soil above the footing is conservatively considered that does not contribute to the stiffness, but exerts its weight with a specific weight of 18 kN/m^3.

2.3 Procedure/Analysis strategy

For representing the effects of the reinforcement, as has already been suggested, the rebar layers are embedded into the concrete.

The effects of the ground are incorporated by means of a horizontal plane with which the concrete of the footing interacts through a subgrade reaction modulus. This allows tracking the areas over which separation between the footing and the ground might be expected to take place.

General contacts are established between the steel flanges and the concrete of the footing.

From the viewpoint of the solution procedure, an explicit integration was preferred, primarily because of the strong nonlinearities of the problem, primarily caused by concrete cracking and crushing. Mass scaling was used in order to increase the time step, thereby increasing the speed of convergence towards the static solution sought.

The calculations involved two successive steps. In a first step the gravity (namely the weight of the footing and that of the overlying soil) was applied; also, when they existed, the pre-stressing loads were also applied in this first step.

Once equilibrium existed under the loads applied in the first step, the external loads were applied in a second step; those were the loads corresponding to the ultimate limit state, which were introduced with a smooth step was used in order to avoid sharp changes in the applied load or its rate of application.

The external loads in the ultimate limit state, namely a shear force and a bending moment, are applied through a distributing coupling on the upper edge of the steel shell embedded in the footing that can be seen in Fig. 3.

3. Results obtained

Having gone through the two steps of load application, Fig. 7 illustrates the evolution of various energies through the loading process. As can be seen there, internal and plastic energies start to grow very gradually with the application of external loads. However, when about 90% of the ultimate loads are applied, both of those energies start to grow much more rapidly and, even more significantly, the kinetic energy is no longer negligible. Failure has taken place and the collapse process is gradually accelerating.

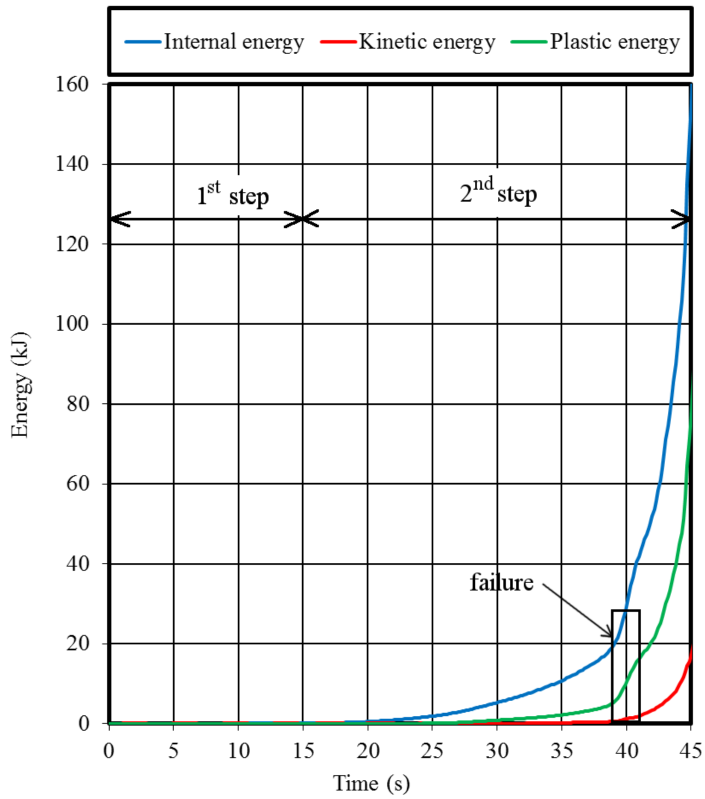

Figure 7. Evolution of energies during the loading process.

The following figures allow observing some of the characteristics of the failure mode being developed. Fig. 8 shows the displacements, duly magnified by a factor of 20 for easier visualization.

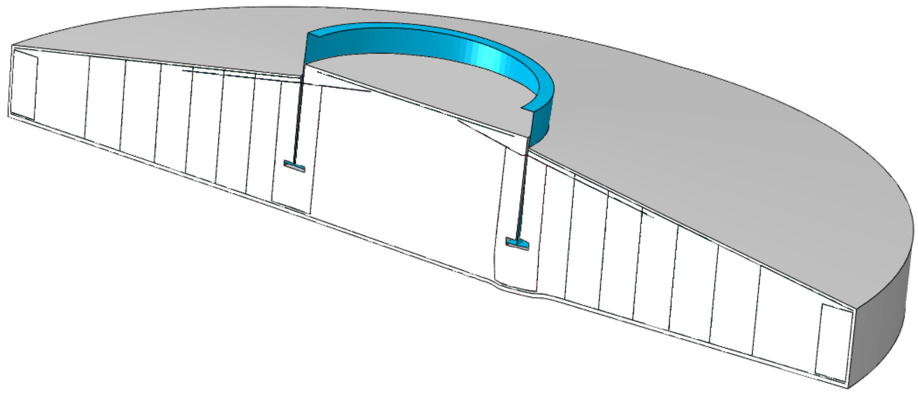

Figure 8. Deformation at failure point (deformation scale x20).

The compression of the concrete, which results from the combination of the applied shear force and bending moment, is depicted in Fig. 9; high values can be seen to concentrate in the upper region of the contact between the internal steel shell and the footing. The figure reflects the situation at the time of failure.

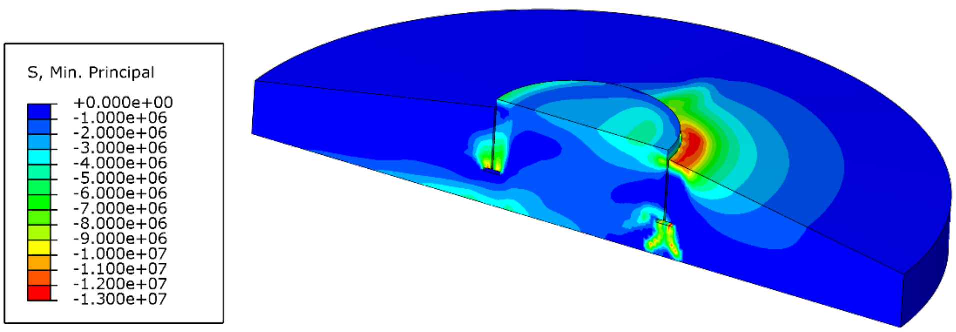

Figure 9. Compression in concrete (Pa).

The effect is perceived, perhaps even better, in Fig. 10, which displays the directions of maximum compression in the concrete, again at the time of failure. Of course the high concrete compression developed in the concrete near the top of the steel shell is corresponded with relatively high tensile stresses directly underneath that steel shell.

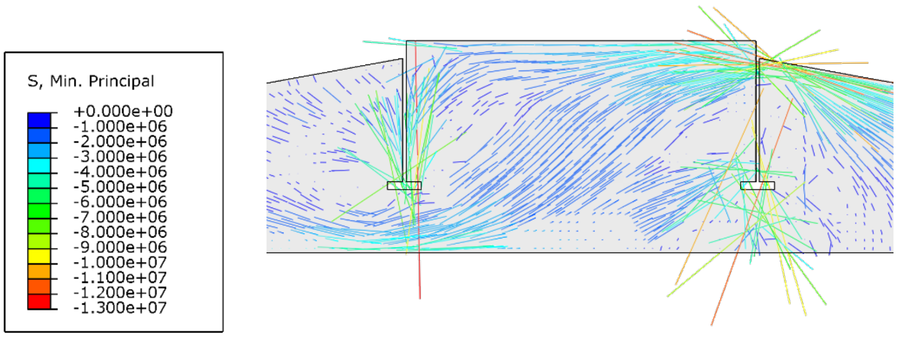

Figure 10. Direction of compressions (Pa).

Indeed the maximum strains developed in the concrete are those reflected in Fig. 11. The figure shows how the tensile strains concentrate in the area mentioned.

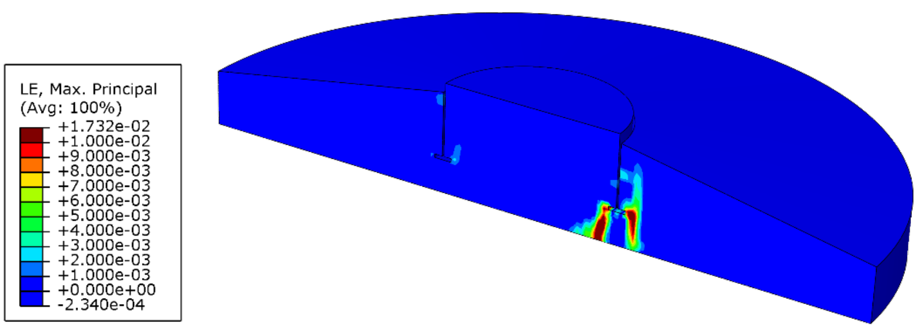

Figure 11. Maximum strains (-).

As a consequence of the tensile demands generated in that region, the reinforcement is severely stressed. The two views included in Fig. 12 clearly manifest the levels and distribution of the stresses developed in the various components of the reinforcement configuration of the footing.

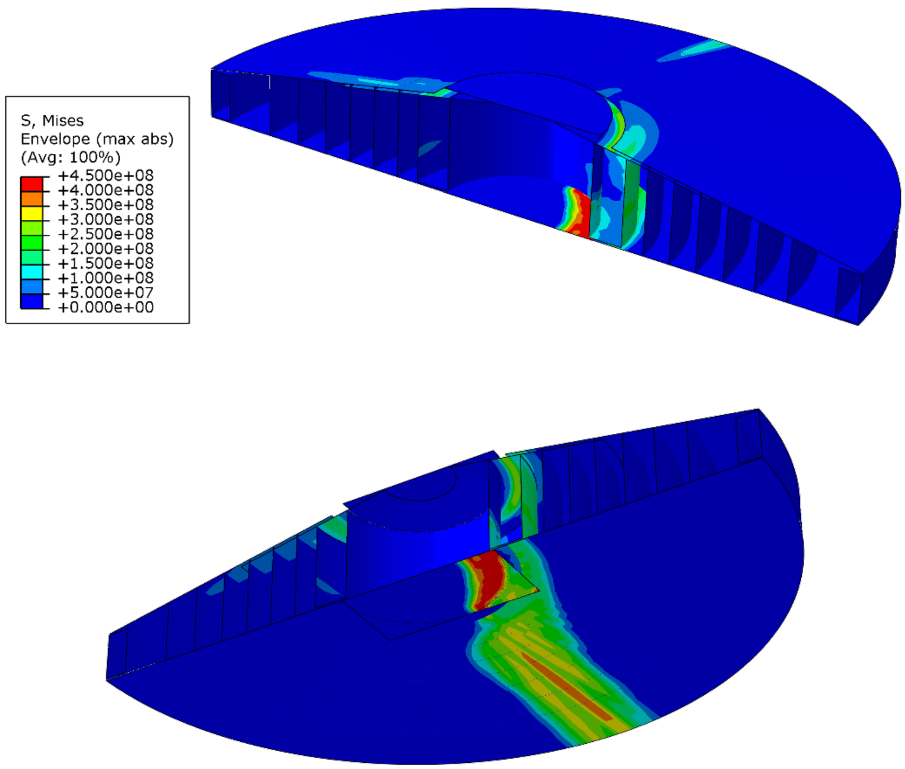

Figure 12. Stresses in the reinforcement (Pa).

Many other types of information can be acquired from the simulations performed. For example, of particular interest for the global stability of the tower on its foundation, Fig. 13 presents the contact pressures developed across the interface between the footing and the ground. It can be seen that, at the failure stage being depicted here, the footing has already separated from the ground over more than the initial contact surface.

As a consequence of the results obtained in the calculations, of which the previous figures provide a concise sample, remedial measures had to be designed for the footing. They involved various modifications as well as the use of post-tensioned bolts. The remedial measures were analysed using procedures similar to those described here in the context of the verification of the original design.

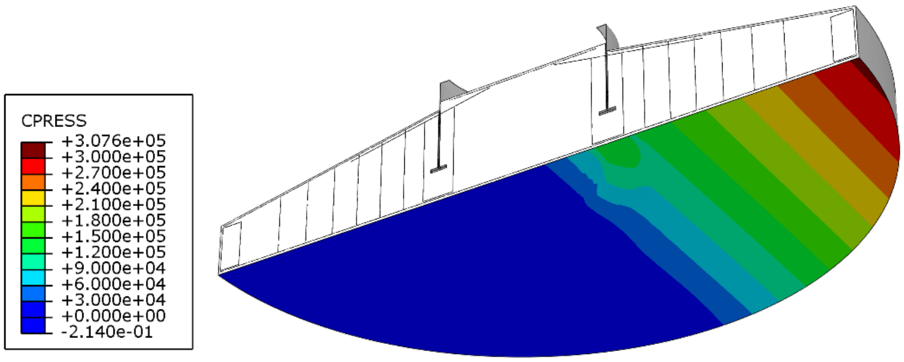

Figure 13. Contact pressures (Pa).

4. Conclusions

A robust methodology has been proposed and applied for dealing with the problems posed by the foundations of wind turbine towers. The concrete geometry is meshed with solid elements. Surface elements are embedded in order to represent the reinforcement layers. Other parts of the structure as the steel cylinder, welded studs, anchorage bars, etc. are explicitly modelled.

From the viewpoint of the mechanical behaviour, concrete is characterised with the Abaqus concrete damaged plasticity model, while traditional elastoplastic descriptions are used for the various steel elements.

The problem is best analysed using the Abaqus/Explicit solver. This is motivated by the strong nonlinearities of the problem, primarily caused by concrete cracking and crushing. Also, it allows easier progress once the failure process starts to accelerate

5. References

1. Abaqus User's Manual, Version 6.14, Dassault Systémes Simulia Corp., Providence, RI.