WARSAW UNIVERSITY OF TECHNOLOGY Index 351733
FACULTY OF CIVIL ENGINEERING
COMMITTEE FOR CIVIL AND WATER ENGINEERING
POLISH ACADEMY OF SCIENCES ISSN 1230-2945

DOI: 10.24425/ace.2020.134422

ARCHIVES OF CIVIL ENGINEERING

Vol. LXVI ISSUE 3 2020

FINITE ELEMENT MODELLING OF THE HEXAGONAL WIRE MESH

M. Grodecki[1]

This paper presents the results of Finite Element Method (FEM) modeling of double-twisted steel hexagonal wire mesh used to construct gabion cages. Gabion cages, filled with soil (usually rock particles) are commonly used in civil engineering (for example in order to form a retaining wall). Static tensile tests are modeled and the obtained force - displacements curves are compared with the laboratory test results (known from literature). Good accordance between numerical and laboratory test results is observed. Three different material models for single wire and double twist are tested. Special attention is paid to double-twist modelling. Simulations of the damaged mesh are also performed, strength and stiffness reduction is analyzed. Anisotropic membrane model for mesh is proposed and calibrated. Parameters for homogenized Coulomb - Mohr media for gabion (filling and mesh) are estimated. Such homogenized Coulomb - Mohr model could be used in engineering practice to model behaviour of real gabion structures.

Keywords: gabion, double-twisted mesh, Finite Element Method (FEM), homogenization, numerical modelling

[1] PhD., Eng., Cracow University of Technology, Faculty of Civil Engineering, ul. Warszawska 24, 34-155 Warsaw, Poland, e-mail: mgrode@pk.edu.pl

1. INTRODUCTION

Steel wire meshes are widely used in civil engineering. One of the typical applications is the formation of a box, filled with soil (usually rock particles), called a gabion. Gabions are used to form constructions such as retaining walls, sound barriers, erosion protection systems (especially in hydrotechnical engineering). These structures are relatively cheap and very efficient. The behavior of gabion components should be first investigated in order to describe engineering behavior of gabion. Behavior of the double-twisted hexagonal wire mesh (presented in Fig. 1) is the main object of investigation in this paper. As it was stated in [4], such mesh is a woven system, which is made by twisting continuous pair of wires three half turns (so-called double twist). Adjacent wires are then connected to form hexagonal openings. Terminal (selvedge) wire is used to edge the mesh. The hexagonal shape improves the macroscopic mesh strength and stiffness. The double-twist prevents mesh from unravelling due to accidental single wire cutting. The most commonly used mesh types are (width x height x wire diameter [mm]): 100x120x2.7, 100x120x3.0, 80x100x2.7, 80x100x3.0 and 60x80x2.7.

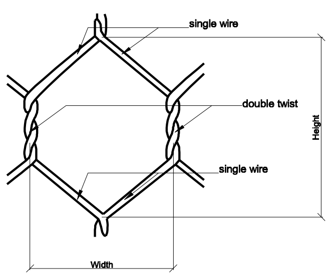


Fig. 1. Basic shape of hexagonal wire mesh

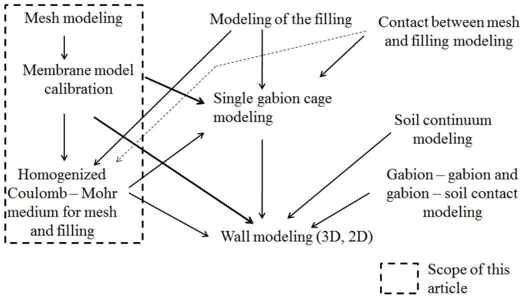


Fig. 2. Scope of this article in the field of gabion modelling

The entire process of the gabion modeling, which is the scope of this article, is presented in Fig. 2. Scope of this article in the field of gabion modelling. Behaviour of the mesh could be investigated during laboratory tests or with the use of numerical simulations. Typical laboratory test is a static tensile test described in [1], [12] and [13]. The published results are often limited to ultimate tensile strength of the mesh and corresponding ultimate strain (elongation) ([1]), usually without any reference to material properties of used steel. More complete set of obtained results is presented in [3], [4], [12] and [13], where force-displacement curves for different meshes are presented along with stress-strain curves for steel. Mesh-soil interaction could also be a matter of investigation ([11]). Single gabion static compression tests are also performed, numerical models are proposed and calibrated ([6]).

Due to complexity of the geometry, use of classic engineering method of static analysis is almost impossible – construction has a high degree of static indeterminacy. Moreover, a double twist is a source of complication – twisted wires are subjected not only to tension (like single wire), but also to bending and shearing. However, a simplified "stress" definition used for double wire (normal force divided by cross-section area) is usually used.

In such a situation numerical analysis could be used to simulate tensile behaviour of the mesh. Numerical model can be calibrated in order to properly replace the laboratory tensile tests during which force-displacement relationship and ultimate load are obtained. Examples of such analyses are presented in [3] and [4]). Also, behaviour of double twist could be a subject of investigation ([10]).

Numerical simulations are performed with the use of Finite Element Method (FEM) ([10]) or Discrete Element Method (DEM) ([3], [4] and [12]).

Building the mesh of numerical model with the use of 3-dimensional continuum elements describing the complex geometry of the structure (especially of the double twist) is almost impossible and numerically ineffective ([10]). Thus, usually simplified model (beam model) describing the behaviour of the wire on the rod level is used. Such model for a single wire could be calibrated on the base of laboratory tensile tests. Laboratory tests can also be conducted for double wire, however, the alternative approach proposed in [10]. (calibrating on the base of 3-dimensional numerical simulation of the double wire tensile test) could also be used.

In general, behaviour of mesh and its components (single and double wire) is strongly nonlinear.

2. NUMERICAL SIMULATIONS OF THE TENSILE TESTS

Static tensile tests described in detail in [4] were simulated. Mesh sample global sizes were Lx=100 cm (perpendicular to the twist) and Ly=50 cm (parallel to the twist). Different mesh sizes (80x100 mm, 100x120mm) and wire diameters (2.7 and 3 mm) were taken into account. First of all, the results were analyzed critically and some discrepancies were observed. For mesh size 100 x 120 mm made of 2.7 mm wire, two experimental force-displacement curves match well, but the third one does not. For mesh of the same size made of wire of different diameter, the ultimate strength should be proportional to the cross-section area of the wire, but for mesh size 80 x 100 mm this is not the case. The described discrepancies show that the proper conduction of the laboratory tensile test of the mesh is a difficult problem itself and the obtained results should be verified.

FEM system ZSoil (described in detail in [5]) was used to perform numerical calculations. Three following material models for single wire and double twist were used. The first model was elastic-plastic model with strain-stress relationship in accordance with [4]. The second model was elastic-ideally plastic (with values of Young modulus E and ultimate strength in accordance with [4] later called the ideal EP. The third model was bi-linear, similar to the model proposed in [10]. The second and third models could be treated as simplifications of the first model. Strain-stress relationships for presented models are presented in Fig. 3. Only normal forces were taken into account. The problem was treated as displacement-driven and large deformations approach was used. The sample numerical model is presented in Fig. 4.

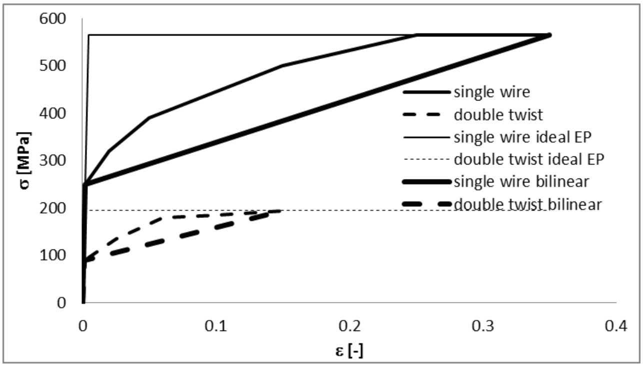


Fig. 3. Constitutive relations for single and double twist wire

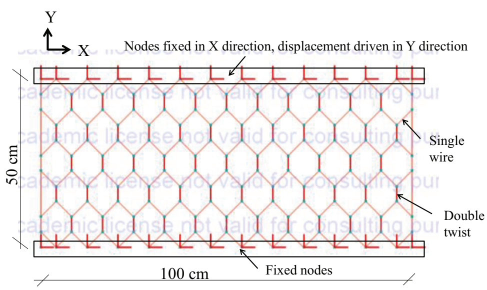


Fig. 4. Numerical model of the mesh with boundary conditions

Simulations of the damaged mesh (with some wires cut) were also performed.

3. OBTAINED RESULTS

In general, good compliance between results of numerical simulations and laboratory test is obtained. The values of ultimate load and force-displacement curves are compared. Numerical tests reveal that mesh is destroyed by double twist failure and this is why the ultimate load of the mesh N could be calculated from a simple formula:

(2.1)
$$N = \frac{\sum_{i=1}^{n} A_i f_{ti}}{L}$$

where:

n – number of wires, A_i – cross section area of a single wire [m^2], f_{ti} – tensile strength of the wire [MPa], L – length of the mesh sample in the direction perpendicular to loading

If all wires are the same (no selvedge wire with higher diameter or mesh sample is large enough to neglect influence of the selvedge wire), the above equation takes even a simpler form:

$$N = \frac{A \cdot f_t}{w} \tag{2.2}$$

where:

w –mesh opening (width)

The values of ultimate load obtained from numerical simulations were compared with laboratory test results and analytical formulas described above (Table 1). Good accordance between values of ultimate load obtained in laboratory tests, numerical simulations and proposed simple formulas show that all used material models properly estimate ultimate load of the structure and proposed analytical formulas could also be used. Proposed analytical formula (Eq. 2.2) gives a bit lower values of ultimate load (in comparison with laboratory tests results) because in the laboratory test selvedge wire with higher diameter was used.

Table 1. Ultimate load of the mesh – results of laboratory tests and numerical simulations

	Ultimate load [kN/m]				
Mesh type	Laboratory (after [4])	Numerical model – elastic-plastic model	Numerical model – ideal EP	Numerical model - bilinear	Proposed formula (Eq. (2.2))
100x120x2.7	33-34	24.9	25.9	23.9	22.4
100x120x3.0	29-48	30.7	32.5	30.6	27.6
80x100x2.7	30-35	29.9	24.7	27.3	27.9
80x100x3.0	51-57	32.5	29.2	30.4	34.5
60x80x2.7	48-52	35.1	34.0	37.5	33.6

The comparison of force-displacements curves obtained from numerical simulations and laboratory tests show that model that uses stress-strain relationship from laboratory test can reproduce macroscopic behaviour of the mesh very well. Elastic-ideally plastic model has tendency to overestimate the stiffness of the mesh while bilinear model – to underestimate. The sample force-displacement curves are presented in Fig. 5.

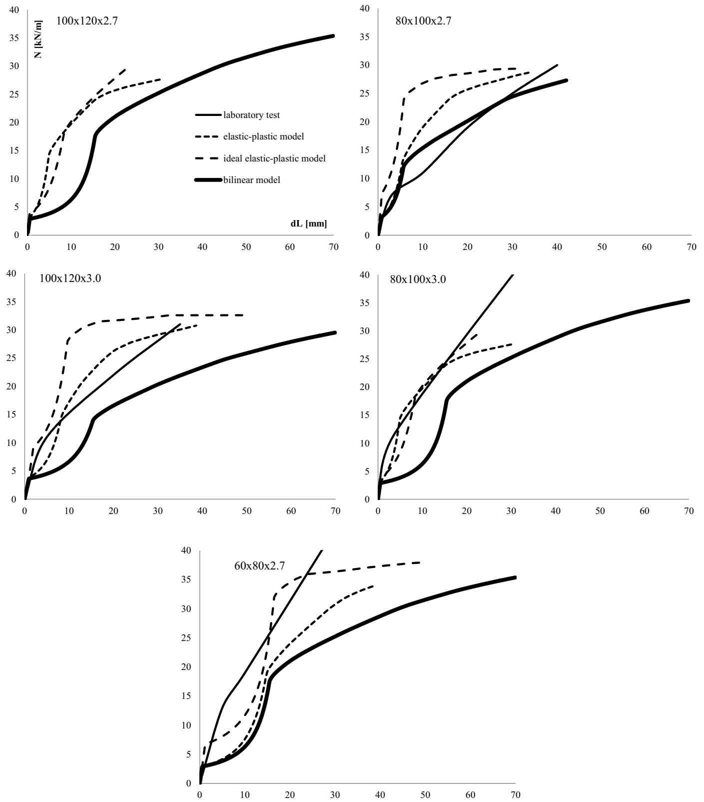


Fig. 5. Force – elongation relationships for different meshes. Comparison of laboratory tests and numerical simulations results

Statistical analysis of the obtained results was performed. Deviation of force N_{dev} defined as

(2.3) $$N_{dev} = \sqrt{\frac{\sum_{i=1}^{n}\left(N_l - N_c\right)^2}{n}}$$

where:

N_l –value od force obtained in laboratory tests

N_c –value od force obtained in numerical calculations

n – number of points on force – elongation curve was calculated for every proposed material model and type of mesh. Obtained values of N_{dev} are presented in Fig. 6.

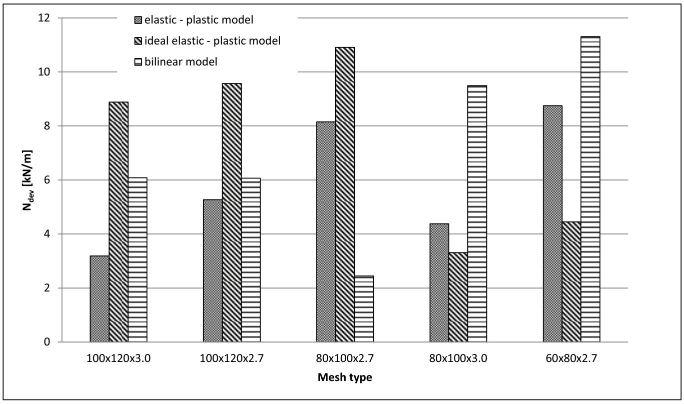


Fig. 6. Values of N_{dev} for different material models and mesh types

For 100x120 type meshes elastic-plastic model gives best correlation between laboratory and numerical results (lowest values of N_{dev}). For 80x120x2.7 mesh bilinear model and for 60x80x2.7 ideal elastic-plastic model gives better correlation.

Simulations of tensile behaviour of the damaged mesh with some wires cut (one, two or three – marked in Fig. 7) show that cutting of one double twist near the centre of the mesh reduces mesh strength by approx. 23%. Cutting two or three double twists does not result in significant strength reduction. However, stiffness of the mesh is significantly reduced after cutting the consecutive wires – as it could be seen in Fig. 8.

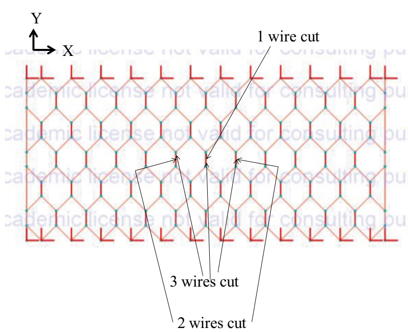


Fig. 7. Cut wires included into damaged mesh behaviour simulation

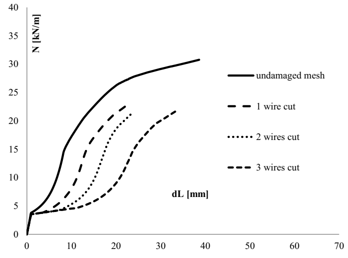


Fig. 8. Force – elongation relationship for damaged mesh size 100x120 mm made of 3 mm wire

4. MEMBRANE MODEL CALIBRATION

Given the numerical model of the mesh consisting of tested and calibrated beams, there could be an attempt to construct a simpler model - membrane model. Anisotropic membrane model with limited tensile strength was used in this article. Such a model has a set of parameters, which need to be calibrated:

Kxx, Kyy – elasticity modulus of the membrane in the X and Y direction [kN/m]

Kxy – shear modulus of the membrane [kN/m]

Ftx, Fty – tensile strength of the membrane in the X and Y direction [kN/m]

Fcx, Fcy – compressive strength of the membrane in the X and Y direction [kN/m]

Kxx and Kyy were calibrated on the base of previously calibrated beam model of the mesh – in order to properly reproduce tensile behavior of the repeatable cell of the mesh. Minimalization of $N_{dev,m}$

$$N_{dev,m} = \sqrt{\frac{\sum_{i=1}^{n}\left(N_{rc} - N_{m}\right)^{2}}{n}} \quad (4.1)$$

where:

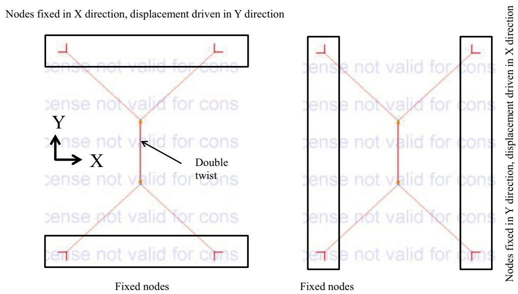
N_{rc} –value od force obtained from repeatable cell model

N_{m} –value od force obtained from membrane model

n – number of points on force – elongation curve

both in Y direction parallel to the twist and X direction perpendicular to the twist was a calibration criterion).

Kxy was set to 0. Fty was estimated with the use of Eq. (2.2), Ftx was estimated as ultimate load of the mesh in the direction perpendicular to the twist (on the base of the repeatable cell of the mesh model), Fcx and Fcy were set to 0. Repeatable cell of the mesh is presented in Fig. 9. Obtained values of membrane parameters for different meshes are presented in Table 2.



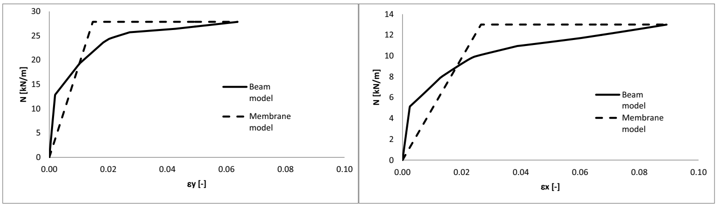
a)tensile test in the Y direction b) tensile tests in the X direction

Fig. 9. Repeatable cell of the mesh

Table 2. Estimated values of membrane model parameters

Mesh type	Kxx [Kn/m]	Kyy [kN/m]	Ftx [kN/m]	Fty [kN/m]
100x120x2.7	398.2	1536.3	10.52	22.55
100x120x3.0	490.3	1890.8	13.00	27.85
80x100x2.7	295.8	1445.4	10.24	25.7
80x100x3.0	365.2	1748.7	12.64	31.7
60x80x2.7	488.8	2362.3	14.00	33.6

The obtained with the use of membrane model load-strain curves were compared with those obtained from beam model of the repeatable cell of the mesh and good correlation was observed (Fig. 10). Also force-elongation curves obtained with use of membrane model, beam model (with elastic-plastic material model) and from laboratory test (after [4]) were compared (Fig. 11). Membrane model has tendency to overestimate stiffness of the mesh but obtained correlation is still satisfactory. Values of N_{dev} (Eq 2.3) were calculated in order to verify correlation between laboratory tests and membrane model results (Fig. 12). Correlation is of course worse than for models described before (Fig. 6) but still satisfactory.



a) load in the twist direction b) load perpendicular to the twist

Fig. 10. Comparison of load – strain curves obtained from beam model of the repeatable cell of the mesh and membrane model (for 100x120 mesh made from 3mm wire)

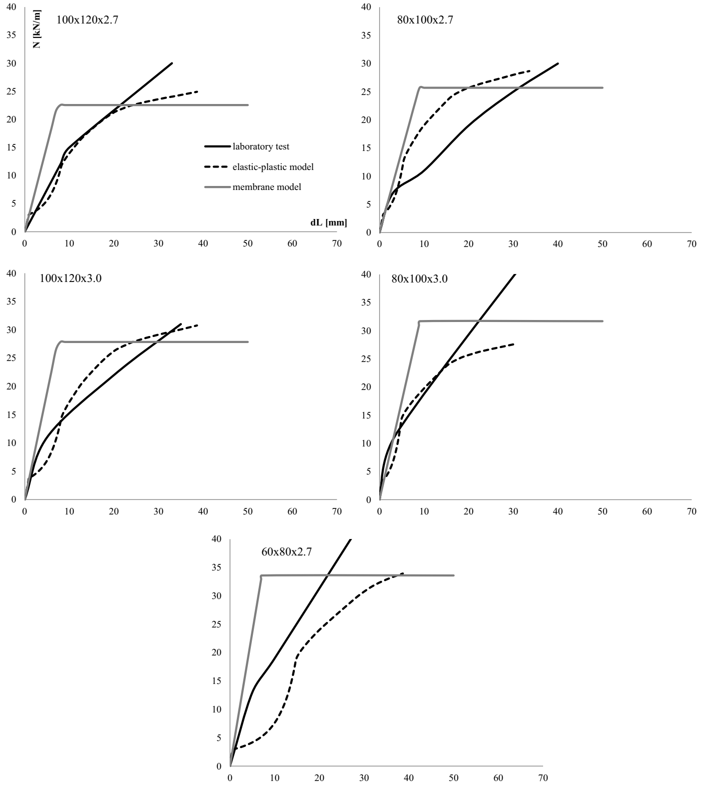


Fig. 11. Force – elongation relationships for different meshes. Comparison of laboratory tests and numerical simulations (elastic-plastic beam model and membrane model) results

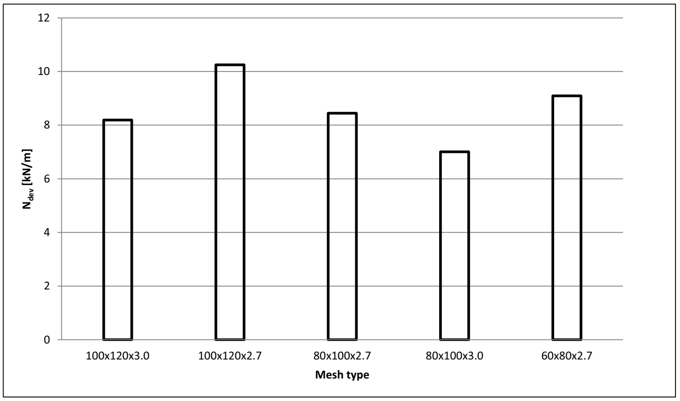


Fig. 12. Values of N_{dev} for membrane model, case of load parallel to the twist

Therefore, membrane model could be used in further research, for example in the whole gabion compression test simulations or in whole gabion wall behaviour simulations.

Estimated values of Kxx, Kyy, Ftx and Fty show that behavior of the mesh is really anisotropic (ratio Kyy/Kxx is between 3.85 and 4.89, Fty/Ftx between 2.14 and 2.51, depending on the mesh type).

5. HOMOGENISED CONTINUUM FOR MESH AND FILLING PARAMETERS

Gabions are often modeled as a homogenized continuum of Coulomb - Mohr type. According to [2] and [9] friction angle of such material is equal to the friction angle of the filling (usually about 40-45^0) and some additional cohesion is used. Such additional cohesion could be calculated from the following formula:

(5.1)
$$c_r = \frac{\Delta\sigma_3}{2}\tan\left(45^0 + \frac{\varphi}{2}\right)$$

where:

c_r – additional cohesion, $\Delta\sigma_3 = \frac{2Kyy \cdot \varepsilon_c}{d} \cdot \frac{1}{(1-\varepsilon_a)}$ – increased confining pressure, $\varepsilon_c = \frac{\left(1-\sqrt{1-\varepsilon_a}\right)}{1-\varepsilon_a}$ - circumferential strain, ε_a – axial strain at steel mesh failure, d – characteristic dimension of the sample (lowest gabion dimension), φ – internal friction angle of the filling

Mesh tensional strength does not appear explicitly in such approach, however, it could be seen that it is a function of axial strain at steel mesh failure and membrane elastic modulus

(5.2)
$$\varepsilon_a = \frac{F_{ty}}{Kyy}$$

The obtained with the use of values of Kyy and Fty, which were estimated before, values of additional cohesion for $\phi=40^0$ and d=0.5 m are summarized in Table 3.

Table 3. Additional cohesion values for different meshes

Mesh type	c_r [kPa]
100x120x2.7	50.0
100x120x3.0	61.5
80x100x2.7	57.4
80x100x3.0	70.8
60x80x2.7	74.4

Presented here approach to a homogenised Coulomb – Mohr type model for gabion could be used in engineering practice for modelling of the gabion retaining walls, examples are presented in [7] and [8].

6. CONCLUSIONS

Presented numerical models (beam and membrane) of the wire mesh are useful tools, which properly describe behavior of the mesh in tension conditions. Thus, membrane model can be used as an element of the whole gabion model (together with appropriate model of filling and interface between the filling and mesh). Required data for model are relatively simple (strain-stress relationship for single wire, double twist and geometry of the mesh). Simplified models (which do not require detailed strain-stress relationships) have limited usability, however, for preliminary approximate calculations give reasonable results. The proposed simple analytical formulas for calculations of the ultimate load of the mesh give results compatible with laboratory tests and numerical simulations, thus they can be used in engineering practice.

REFERENCES

1. R. Agostini, L. Cesario, A. Conte, M. Masetti, A. Papetti "Flexible Gabion Structures In Earth Retaining Walls". Officine Maccaferri, Bolonia, 1987
2. R. J. Bathurst, K. Rajagopal "Large-scale triaxial compression testing of geocell reinforced granular soils", Geotechnical Testing Journal vol. 16 no. 3: 296-303, 1993
3. D. Bertrand, F. Nicot, P. Gotteland, S. Lambert "Modelling a geo-composite cell using discrete analysis", Computers and Geotechnics 32: 564-577, 2005
4. D. Bertrand, F. Nicot, P. Gotteland, S. Lambert "Discrete element method (DEM) numerical modeling of double-twisted hexagonal mesh", Canadian Geotechnical Journal 45(8): 1104-1117, 2008
5. S. Commend, S. Kivell, R. Obrzud, K. Podleś, A. Truty, T. Zimmermann "Computational Geomechanics On PC". Rossolis Editions, Lausanne, 2016
6. L. Der-Guey, H. Bor-Shun, L. Shin-Hwei "Deformation analyses of gabion structures" In: Proceedings of Interpraevent, Taiwan, 2010
7. M. Grodecki "Failure modes of gabion retaining walls", Technical Transactions 24: 55-62, 2015
8. M. Grodecki, A. Urbański "Landsliding slope supported by gabions – a case study and the methodology of numerical modelling", Technical Transactions 12: 53-59, 2018
9. P.K. Jayasree "Performance of gabion faced reinforced earth retaining walls". PhD Dissertation, Cochin University of Science and Technology, 2008
10. S. Śpiewak "The modeling and strength analyzing of gabion grids", Górnictwo Odkrywkowe 4–5/2014: 177-182, 2014 [in Polish]
11. C. Teerawattanasuk, D. Bergado, W. Kongkitkul "Analytical and numerical modeling of pullout capacity and interaction between hexagonal wire mesh and silty sand backfill under an in-soil pullout test", Canadian Geotechnical Journal 40: 886-899, 2003
12. K. Thoeni, C. Lambert, A. Giacomini, S.W. Sloan "Discrete modelling of hexagonal wire meshes with a stochastically distorted contact model", Computers and Geotechnics 49: 158-169, 2012
13. G. Yang, X. Huang, Y. Lin Y "Test study on engineering properties of gabion structures" In: Proceedings of International Symposium on Geoenvironmental Engineering ISGE2009, Hangzhou, China, 2009

LIST OF FIGURES AND TABLES:

Fig. 1. Basic shape of hexagonal wire mesh / Rys. 1. Sześciokątna siatka splatana
Fig. 2. Scope of this article in the field of gabion modelling
Rys. 2. Tematyka artykułu w obszarze modelowania gabionów
Fig. 3. Constitutive relations for single and double twist wire
Rys. 3. Zależności konstytutywne dla pojedynczego i podwójnego skręconego drutu
Fig. 4. Numerical model of the mesh with boundary conditions
Rys. 4. Model numeryczny siatki wraz z warunkami brzegowymi
Fig. 5. Force – elongation relationships for different meshes. Comparison of laboratory tests and numerical simulations results
Rys. 5. Zależność siła – wydłużenie dla różnych siatek. Porównanie wyników laboratoryjnych i symulacji numerycznych
Fig. 6. Values of N_{dev} for different material models and mesh types
Rys. 6. Wartości N_{dev} dla różnych modeli materiałowych i siatek
Fig. 7. Cut wires included into damaged mesh behaviour simulation
Rys. 7. Przecięte druty uwzględnione w symulacjach zachowania się uszkodzonej siatki
Fig. 8. Force – elongation relationship for damaged mesh size 100x120 mm made of 3 mm wire
Rys. 8. Zależność siła – wydłużenie dla uszkodzonej siatki 100x120 mm wykonanej z 3 mm drutu
Fig. 9. Repeatable cell of the mesh
Rys. 9. Powtarzalna komórka siatki
Fig. 10. Comparison of load – strain curves obtained from beam model of the repeatable cell of the mesh and membrane model (for 100x120 mesh made from 3mm wire)
Rys. 10. Porównanie zależności obciążenie – odkształcenie uzyskanych z modelu prętowego i membranowego (dla siatki 100x120 mm wykonanej z 3 mm drutu).
Fig. 11. Force – elongation relationships for different meshes. Comparison of laboratory tests and numerical simulations (elastic-plastic beam model and membrane model) results
Rys. 11. Zależność siła – wydłużenie dla różnych siatek. Porównanie wyników laboratoryjnych i symulacji numerycznych (model pręta sprężysto – plastycznego i model membranowy)
Fig. 12. Values of N_{dev} for membrane model, case of load parallel to the twist
Rys. 12 Wartości N_{dev} dla modelu membranowego, przypadek obciążenia równoległego do splotu
Tab. 1. Ultimate load of the mesh – results of laboratory tests and numerical simulations.
Tab. 1. Nośność graniczna siatki – wyniki badań laboratoryjnych i symulacji numerycznych
Tab. 2. Estimated values of membrane model parameters
Tab. 2. Oszacowane wartości parametrów modelu membranowego
Tab. 3. Additional cohesion values for different meshes
Tab. 3. Wartości dodatkowej kohezji dla różnych siatek

MODELOWANIE NUMERYCZNE SZEŚCIOKĄTNEJ SIATKI SPLATANEJ

Słowa kluczowe: gabion, siatka splatana, Metoda Elementów Skończonych (MES), homogenizacja, modelowanie numeryczne

STRESZCZENIE

Artykuł przedstawia wyniki symulacji numerycznych (wykonanych z wykorzystaniem Metody Elementów Skończonych MES) zachowania się stalowej splatanej sześciokątnej siatki używanej do budowy gabionów. Gabiony, wypełnione gruntem (najczęściej gruboziarnistym) są powszechnie używane w budownictwie, np. do budowy murów oporowych. Modelowano testy statycznego rozciągania siatki, uzyskane zależności siła – przemieszczenie porównano z literaturowymi wynikami badań laboratoryjnych. Zaobserwowano dobrą zgodność pomiędzy wynikami laboratoryjnymi a rezultatami symulacji numerycznych. Testowano trzy różne modele konstytutywne dla pojedynczego i podwójnego splecionego drutu. Specjalną uwagę poświęcono modelowaniu podwójnie splecionego drutu. Wykonano też symulacje zachowania się uszkodzonej siatki, analizowano spadek jej nośności i sztywności. Zaproponowano i wykalibrowano anizotropowy model membranowy dla siatki. Oszacowano parametry zhomogenizowanego ośrodka typu Coulomba – Mohra dla gabionu (siatki i wypełnienia). Model ten może być wykorzystywany w praktyce inżynierskiej do modelowania rzeczywistych konstrukcji gabionowych.

Received: 07.05.2020 Revised: 02.07.2020