ISSN 1392-1134/ eISSN 2345-0371
Agricultural Engineering, Research Papers, 2015, Vol. 47, 1-5 p.

WEAR MODEL DEVELOPMENT OF SOIL TILLAGE ELEMENT

R. Skirkus, V. Jankauskas
Aleksandras Stulginskis University, Lithuania

In the last ten years computer technology and fast development of numerical methods enabled numerical modelling by applying the finite element method (FEM). For our research we selected SIMULIA ABAQUS because it has huge library, detailed documentation, user friendly interface, many built in functions. In this study we have modelled two bodies: soil material and the wearing element of cultivator part. We chose the so called Coupled Eulerian Lagrangian method for solving our problem. This method captures the advantages of the Lagrangian and the Eulerian methods. Developed model shows realistic results because observed contact pressure zones match the same regions on weared cultivator element – the highest wear where highest contact pressure. The highest contact pressure was observed on cultivator element front edge bottom corner – 19 MPa.

Keywords: wear model, soil model, cultivator.

Received 2015-10-23, accepted 2015-12-15

INTRODUCTION

One of the most intensive wear types is abrasive wear. Abrasive wear resistant material properties were investigated by known scientists Khrushchov, Babichev, Blickensderfer, Archard, Wahl [1,2,3,4]. The main abrasive wear laws were formulated by M. M. Khrushchov [1].

In the last ten years computer technology and fast development of numerical methods enabled numerical modelling by applying the finite element method (FEM). FEM are used for theoretical simulation of contact and wear to replace the laboratory or high cost field tests. There are many FEM software packages, but most popular software packages for modeling such type of problem are SIMULIA ABAQUS, LS – DYNA, ANSYS. Each software has its strong and weak points like laboratory methods has.

We selected SIMULIA ABAQUS for our research because it has huge library, detailed documentation, and user friendly interface, many built in functions.

EXPERIMENTAL

Soil is a multiphase porous material that consists of mineral particles of different size and shape (solid phase) and voids which are filled with one or more fluids (fluid phase). The mineral particles build up the soil skeleton whose behaviour can be described by various constitutive models depending on the investigated target. In typical geotechnical problems the voids are filled with water and air [5, 6].

Agricultural soils experience plastic deformations after yielding induced by an engaging tillage tool. The behaviour of the soil before yielding is elastic, but the elastic range is usually small and does not represent the realistic stress range [7].

For the problems such ploughing or drilling in agriculture the soil-structure interface experiences large deformations. The frictional contact between soil and structure becomes complicated. The soil can be separated from the structure and the gaps may be reclosed later. When we want to simple solve problem with typical FEM model we get element distortion of soil. Thus, some numerical methods, such as Arbitrary Lagrangian-Eulerian method (ALE), Coupled Eulerian-Lagrangian method (CEL) and Smoothed Particle Hydrodynamics (SPH), which can overcome the mesh distortion problem, have been implemented in Abaqus to solve boundary value problems [5].

The so called Coupled Eulerian Lagrangian method was chosen for solving our problem. This method captures the advantages of the Lagrangian and the Eulerian methods. The Lagrangian and Eulerian methods differently describes the movement of small volumetric fraction [8].

In numerical analyses using this CEL method the Eulerian material is tracked as it flows through the mesh by computing its Eulerian volume fraction. Contact between Eulerian materials and Lagrangian materials is enforced using a general contact that is based on a penalty contact method [9].

In this study we have modelled two bodies: soil material and the wearing element of cultivator part. A box 1 m long, 0,05 m wide and 0,06 m deep was simulated as the soil material. The soil was modelled as Eulerian body. Both, the elastic and plastic properties must be considered when defining the soil material in a FE model and predict the yield and plastic strain behaviour using initial models.

The Drucker–Prager model is a modified version of von Mises model, considering the influence of hydrostatic pressure in failure. The extended Drucker–Prager models can be used to simulate frictional materials, such as rock, soils, where material yield is associated with hardening (i.e. the material strength increases with increasing stress level) [9].

Detailed parameters used in the model for soil are given in tables 1 and 2. It was selected that: soil elastic type – isotropic, density distribution – uniform. Drucker Prager shear criterions were linear and hardening behaviour type – compression.

Table 1. Soil model parameters [10].

Basic parameters:	Value
Density (kg/m^3)	1600
Young's modulus (MPa)	4,46
Poison ratio	0,342
Drucker Prager parameters:	
Angle of friction	30
Flow stress ratio	1
Dilation angle	20

Table 2. Drucker Prager hardening parameters [10].

Yield Stress (MPa)	Abs plastic Strain
13,0	0
20,0	0,0007
24,0	0,001
37,5	0,002
72,5	0,0034
160,0	0,05

Table 3. Johnson – Cook plastic parameters of the cultivator part [11]:

A – initial yield stress (MPa)	556
B – hardening modulus (MPa)	600
n – work-hardening exponent	0,234
C – coefficient dependent on the strain rate	0,013
m – thermal softening	1,0
Reference strain rate	0,001
T_{melt} – melting temperature	1460

The wearing element of cultivator part was modelled as Lagrangian body. For this part also it was defined elastic, plastic and damage parameters. In this model it was used typical parameters of known material steel AISI 1045. It was selected that: metal elastic– isotropic and plastic – Johnson – Cook hardening type, density distribution – uniform.

Abaqus provides a dynamic failure model specifically for the Johnson-Cook plasticity model, which is suitable only for high-strain-rate deformation of metals. This model is referred to as the "Johnson-Cook dynamic failure model." Abaqus also offers a more general implementation of the Johnson-Cook failure model as part of the family of damage initiation criteria, which is the recommended technique for modeling progressive damage and failure of materials. The Johnson-Cook dynamic failure model is based on the value of the equivalent plastic strain at element integration points. The failure is supposed to occur when the damage parameter exceeds 1 [9].

Table 4. Elastic and damage parameters of the cultivator part [11]:

Basic parameters:	Value
Density (kg/m^3)	7870
Young's modulus (GPa)	206
Poison ratio	0,29
Johnson-Cook damage constants:	
D1	0,06
D2	3,31
D3	-1,96
D4	0,0018
D5	0,58

The use of the Johnson-Cook dynamic failure model requires use of hardening parameters but does not necessarily require the use of strain rate dependence. These limitations does not have the described Johnson-Cook damage initiation criterion [9].

The interaction contact properties was set to penalty method with friction coefficient of 0,3 [12]. The penalty contact method searches for edge-into-edge and node-into-face penetrations in the current configuration. The penalty stiffness that relates the contact force to the penetration distance is chosen automatically by Abaqus to get minimal effect on the time increment and the penetration is not significant [9].

The boundary condition was used for soil material box in FE model: the box bottom was fixed and all degrees of freedom were set to zero. For cultivator element part two boundary conditions where set: 1) in type displacement/rotation set that cultivator element parts can't move in y and z directions, also there was limited rotations to zero on all x, y and z axes; 2) in type velocity for cultivator part there was set speed in x axis 2.8 m/s, what its real working speed of the cultivator.

The Eulerian body was meshed with hex EC3D8R element type, which is an 8-node linear Eulerian brick. The Lagrangian body was meshed with structured hex C3D8R element type, which is an 8-node linear brick. This is a commonly used element type for 3D stress–strain analysis of continuum material [9]. The element deletion for Lagrangian mesh was enabled.

RESULTS AND DISCUSION

The modelling was performed on Windows 7 (64 bit) server with Intel Xeon 2.0 GHz 24 cores CPU and 8 GB of RAM. Our model was calculated in 8 hours and 44 minutes. In figure 1 is shown modelled bodies start position.

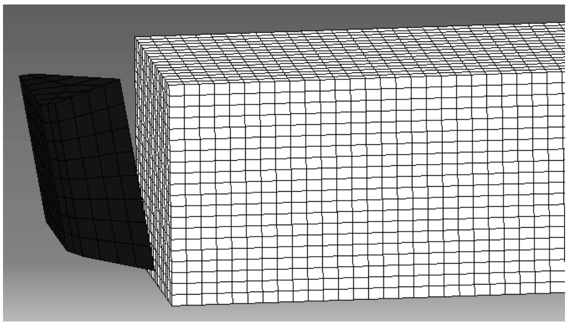
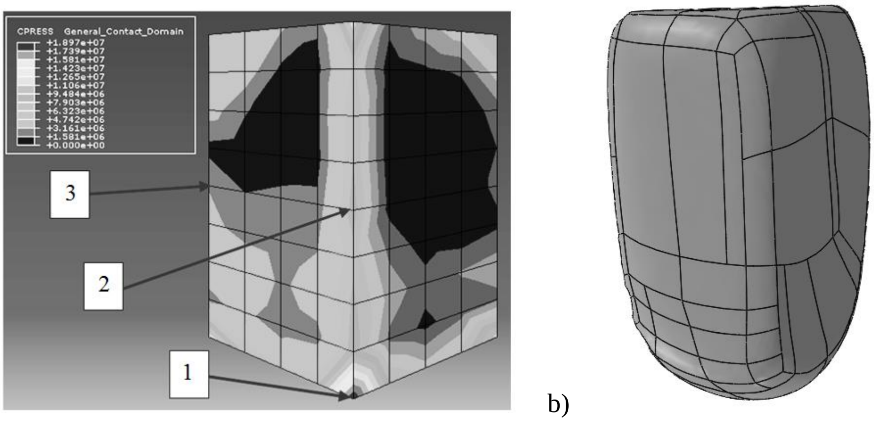


Figure 1. Side view of modelled bodies (left side in blue cultivator element and right its soil box).

In this model was performed 0,84 m displacement of cultivator element part in soil medium. View of cultivator part affected by contact pressure of soil after calculations presented in figure 2.

From figure 2a we see that our modelled cultivator element have biggest contact stress in those places of edges where was the highest wear. The highest wear we can see from picture (figure 2b) where cultivator tips where tested in real applications [13].

a) 

b)

Figure 2. Cultivator element: a - affected by contact stress of soil; b - real, scanned after 46 ha of work.

On the tip there was marked three points (figure 2a) to get numerical values of contact pressure. Those values it's shown in the figure 3.

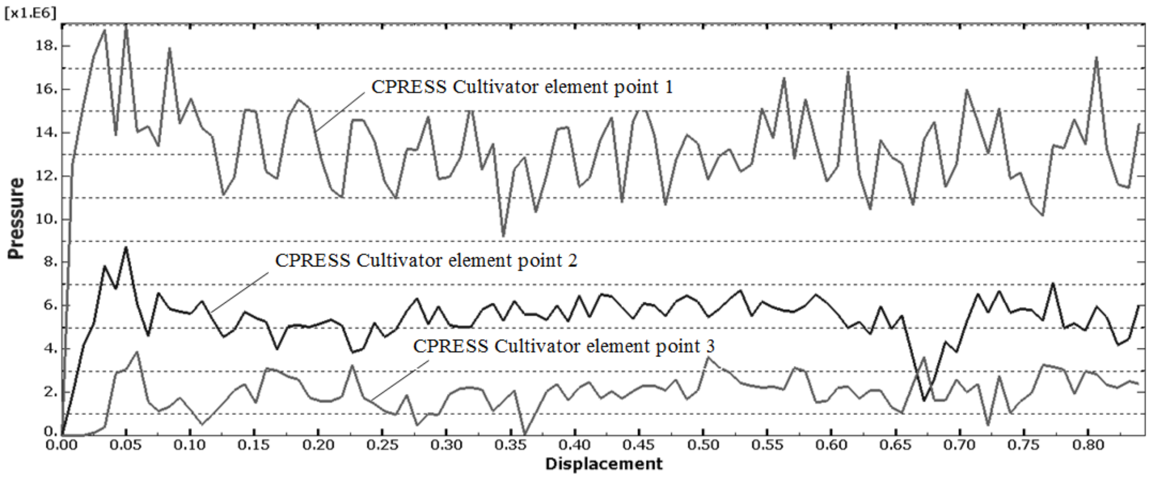


Figure 3. Values of contact pressure in various points of cultivator element.

The highest contact pressure was expected to get on point 1 and there it was the highest. At the beginning only in 0,03 m it was reached maximum value of 19 MPa, but after 0,1 m the contact pressure at point one drops down and stabilizes between 11–15 MPa. It's explained by the simple physical law – at the distance of 0,03 m the static friction force transforms to sliding friction. The same law it's valid and for point number two and three. For those points there was also the static friction transforms to sliding in distance of 0,03–0,04 m. In point two the maximum pressure reaches 9 MPa and then stabilizes between 5–7 MPa. At point three there was lowest observed value, where maximum was 4 MPa and then it plays between 1–3 MPa.

CONCLUSIONS

In this study, a 3D element model for cultivator element – soil was developed. From the analyses carried out the following conclusions can be drawn:

1. The highest contact pressure was observed on cultivator element front edge bottom corner – 19 MPa.
2. Developed model shows realistic results because observed contact pressure zones match the same regions on weared cultivator element – the highest wear where highest contact pressure.
3. The FEM simulation in general is a useful tool to study the complicated soil – tool interactions. The model developed in this study can be employed to design cultivator element and investigate the effects of operational parameters in soil.

4. Developed model can be as basics to develop cultivator element abrasive wear phenomena in further research.

REFERENCES

[1] M.M.Khrushchev, M.A.Babichev, Abrasive wear, Moscov: Nauka, 1970, -252 p. (in Russian).

[2] R.Blickensderfer, G.A. Laird. Pinion–drum abrasive wear test and comparison to other pin tests. J. Test. Eval. 16, 1988, p.516–526.

[3] J.F.Archard, Elastic deformation and the laws of friction. Proceedings of Royal Society of London A, Vol. 243, 1957, p.190–205 (). DOI:10.1098/rspa.1957.0214

[4] H. Wahl, Verschleissprobleme im Braunkohlenbergbad, Braunkohle Wärme Energ., Vol 5/6, 1951, p75-87.

[5] T.Pichler, T.Pucker, T.Hamann, et al, High-Performance Abaqus Simulations in Soil Me- chanics Reloaded – Chances and Frontiers, 2012 SIMULIA Community Conference, 2012, p.1–30.

[6] B.Schümann, Modeling of soils as multiphase-materials with Abaqus, 2010 SIMULIA Customer Conference, 2010, p.1–15.

[7] A.M.Mouazen, M.Neményi, Finite element analysis of subsoiler cutting in non-homogeneous sandy loam soil. Soil and Tillage Research 51 (1–2), 1999, p. 1–15. DOI:10.1016/S0167-1987(99)00015-X

[8] S.Henke, T.Hamann, J.Grabe, Coupled Eulerian-Lagrangian Simulation of the Deep Vibration Compaction Process as a Plastodynamic Problem, Proceedings of the 8th International Conference on Structural Dynamics, EURODYN 2011, 2011, p.4–6.

[9] ABAQUS, ABAQUS User’s Manuals Version 6.13, 2014.

[10] S.N.Mokhatar, R.Abdullah, Computational Analysis of Reinforced Concrete Slabs Subjected to Impact Loads. International Journal of Integrated Engineering 4 (2), 2012, p.70–76.

[11] C.Duan, T.Dou, Y.Cai, Y.Li, Finite element simulation and experiment of chip formation process during high speed machining of AISI 1045 hardened steel. Int. J. on Production and Industrial Engineering 2 (1), 2009, p.28–32.

[12] M.Naderi-Boldaji, R.Alimardani, A.Hemmat, et al., 3D finite element simulation of a single-tip horizontal penetrometer–soil interaction. Part II: Soil bin verification of the model in a clay-loam soil. Soil and Tillage Research 144, 2014, p. 211–219. DOI:10.1016/j.still.2014.03.008

[13] V.Jankauskas, E.Katinas, R.Skirkus, V.Alekneviciene, The method of hardening soil rippers by surfacing and technical-economic assessment. Journal of Friction and Wear 35 (4), 2014, p.270–277. DOI:10.3103/S106836661404014X

Author for contacts:
Remigijus Skirkus
Institute of Power and Transport Machinery Engineering, Aleksandras Stulginskis University, Lithuania.
Phone: +370 68210863, E-mail: remigijus.skirkus@yahoo.com.