3D Finite element simulation of metal waste compaction process

S. Bokhors [1], A. Benhizia [1*], T. Outtas [1]

1 Laboratory of Structural Mechanics and Materials, Faculty of Technology, University of Batna 2

………………………..

benhiziauniv@yahoo.fr

***Abstract*— This paper presents a numerical study of the dynamic compaction of random cylindrical metal waste assembly. The analysis of the process is based on a new method to place the waste randomly in the chamber, using the 3D collision detection function in SolidWorks that can highlight interferences and collisions.**

The material is considered to be elasto-plastic. A three-dimensional study was carried out using the finite element method in an explicit dynamic analysis. The study offer useful information on the packing density variation within the loading applied. The results provide a better understanding of the dynamic compaction process.

Keywords—Elastoplastic, Random, Waste metal, Compaction

I. Introduction

In recent years, interest in dynamic compaction methods of metal has increased due to the need to improve compaction properties and to increase production rates of compacts.

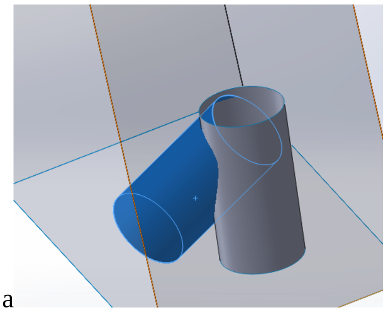
In the literature, there have been many experimental investigations of machining chips compaction as part of metal waste recycling effort (e.g. [1, 2, 3]). However, little exists on the numerical modeling of compaction. Relevant studies have been focused on the compaction of powders. Computational modeling of compaction of powders has been applied using two methods: the discrete model and the continuum model methods. In the discrete model method, powder particles are modeled as individual uniform spheres (in 3D) or circular cylinders (in 2D) and the contact interaction and deformation of the particles are analyzed [4, 5, 6, 7], whereas in the continuum model method the collection of powders is modeled as a continuous media whose deformation with a changing density is analyzed [8, 9, 10, 11, 12].

The aim of the present study is modeling the dynamic compaction process of metal waste using the finite element method and the commercial numerical modeling software ABAQUS, based on several parameters: contact, motion of assembly components in a realistic way, nonlinear properties of the material, boundary conditions and resolution.

II. SIMULATION METHOD AND CONDITIONS

A. Geometry and model

In order to place waste randomly in the box, by a realistic manner, a new technique is proposed basing on the 2D and 3D collision detection function in SolidWorks software, which the interferences, collisions between components can be modeled correctly, when a component is dragged, this one applies a force to components that it touches, and moves the components if they are free to move. (See Fig. 1).


a

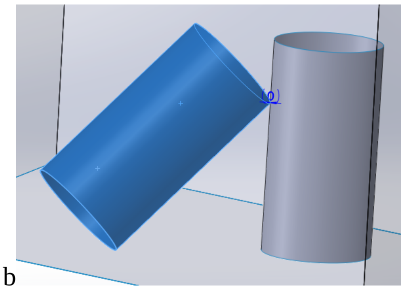

b

Fig. 1. Modeling with: (a) Collision between components ; (b) Detection and stop collision between components.

The Fig. 2 shows an assembly of 45 pieces of metal waste randomly positioning into cubic box.

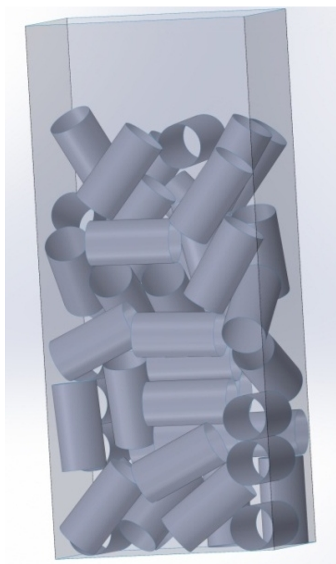


Fig. 2. Three-dimensional model: Assembly of 45 pieces of metal waste ranodmly positioning in the box.

B. Loading process

The compaction process is investigated by a confined compression in an explicit dynamic analysis. The geometry under the uniaxial compression is presented in Fig. 3. The box is constrained at the bottom surface, and a load is applied to the top surface.

The axial loading was achieved by controlling the movement of topwall in x-direction, a pre-determined total x-direction displacement is applied to the topwall.

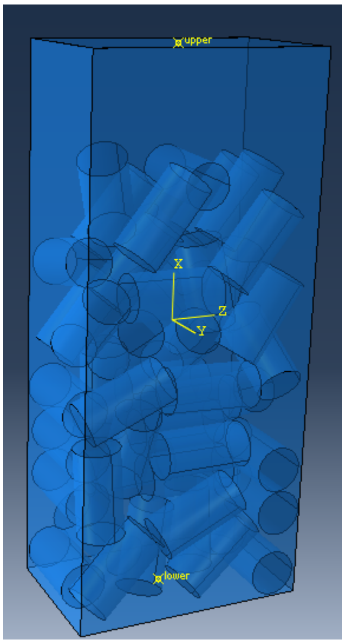


Fig. 3. 3D FE model and boundary conditions applied.

The topwall and the box were considered to be rigid and modeled as an analytical rigid surfaces, attached to the rigid body reference nodes named "upper" and "lower" respectively (See Fig. 3).

A general contact option in ABAQUS was used, applying hard contact in normal direction and frictionless in tangential direction. Those contact conditions avoid the cylinders penetration into the rigid walls and vice versa.

C. Material properties

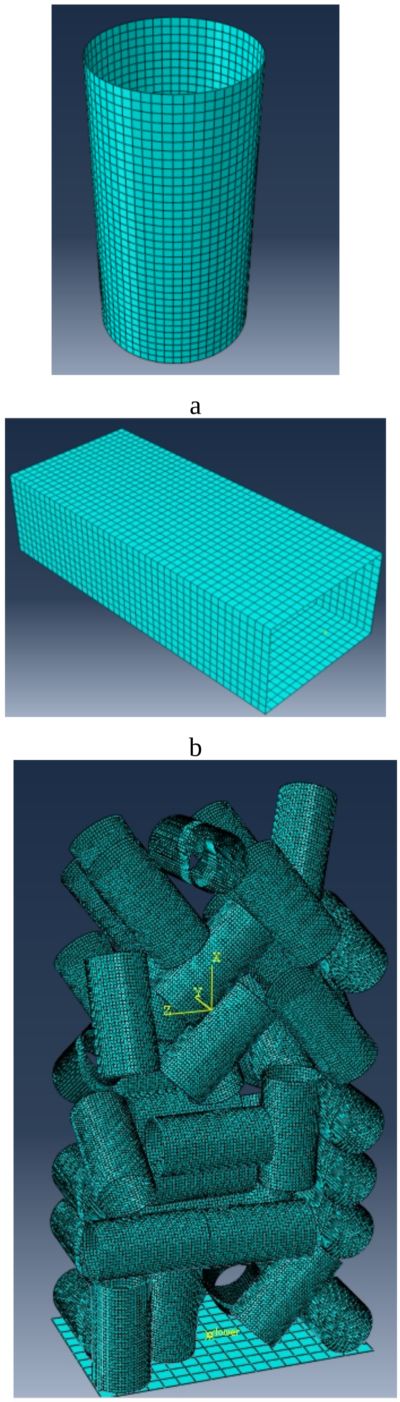
The Metal sheets used in this study are made of aluminum alloys, the material is assumed to be isotropic and elasto-plastic, with a Young's Modulus of E=68GPa and Poisson ratio of v=0.3 [13]. The Plastic characteristics of the tested material are given in table 1.

TABLE I. PLASTIC CHARACTERISTICS.

Plastic characteristics	
Stress [MPa]	***Strain***
80	*0*
115	*0.024*
139	*0.04*
150	*0.079*
158	*0.089*
167	*0.124*
171	*0.149*
173	*0.174*

D. 3D finite element mesh

The global model is meshed with quadrilateral shell elements as shown in Fig. 4, using the S4R element type. This element is a fully integrated general purpose conventional shell element in ABAQUS and it accounts for finite membrane strains and arbitrarily large rotations, which makes it suitable for large-strain analysis [14, 15]. The S4R element is a four-noded element. Each node has three displacement and three rotation degrees of freedom. Each one of the six degrees of freedom uses an independent bilinear interpolation function. The selected element type uses a reduced integration to form the element stiffness with only one integration location per element [15].



c

Fig. 4. 3D FE meshed model: a) Cylindrical waste ; b) Box; c) Assembly of 45 pieces.

III. SIMULATION RESULTS

The results of deformed shape and Von Mises stresses are shown at instance of 0.035 sec (Fig. 5 and Fig. 6), 0.0433 sec (Fig. 7 and Fig. 8), and 0.0483 sec (Fig. 9 and Fig. 10).

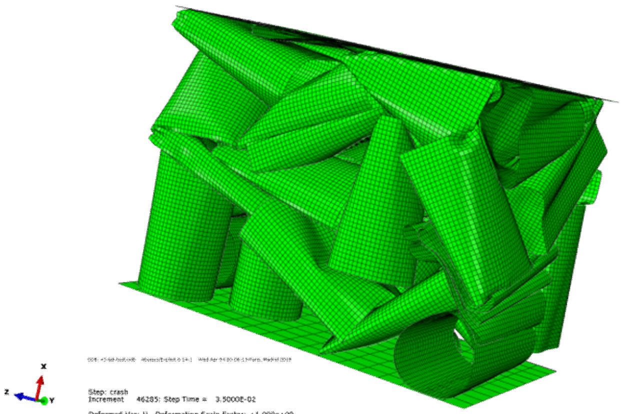


Fig. 5. Deformed shape, t= 0.035 sec.

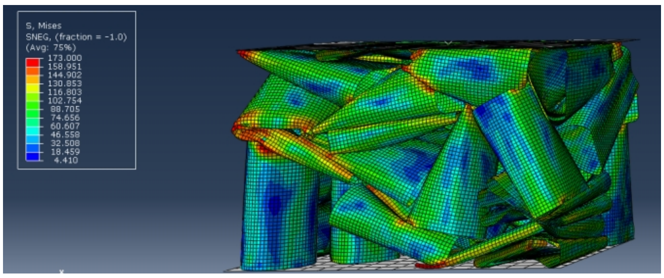


Fig. 6. Von Mises stress, t= 0.035 sec.

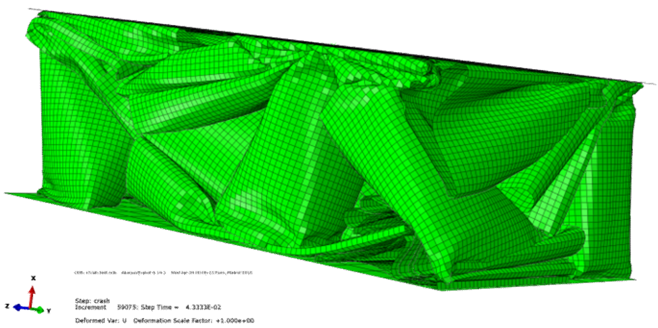


Fig. 7. Deformed shape, t= 0.0433 sec.

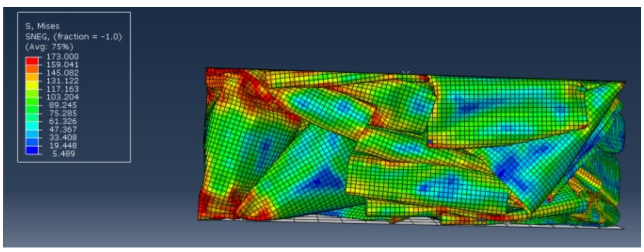


Fig. 8. Von Mises stress, t= 0.0433 sec.

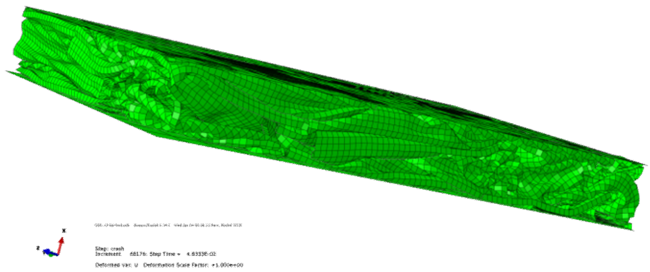


Fig. 9. Deformed shape, t= 0.0483 sec.

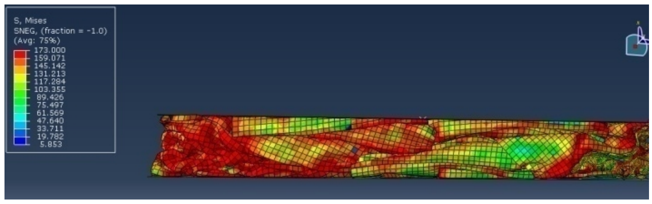


Fig. 10. Von Mises stress, t= 0.0483 sec.

The obtained results according to the material studied, show that:

- During the axial loading process, relative position of cylinders adjusted constantly.
- Relative movement between the cylinders includes not only sliding, but also rotation, once the inter-cylinders force between the cylinders exceeds their contacting strength, relative movements would occur, and the pore space between cylinders would be filled or enclosed by the displaced of this one. The skeleton of the assembly would turn to a more steady state.
- With the increase of the loading rate, the assembly volume decreases.
- The simulation is performed until the target distance is reached. At this point, the pressure and the density values can be obtained.

IV. CONCLUSION

In this study, a three-dimensional model of random cylindrical waste metal assembly is modeled and analyzed under uni-axial compression.

In particular the evolution of density variation within the loading applied was investigated.

To this end, an explicit dynamic analysis was used to simulate the compression process. The simulation is based on the collision detection, where the interferences, collisions between components modeled correctly.

From the obtained results, the controlled pre-determined displacement of the topwall in the axial loading allows to determine the desired density.

The numerical model illustrates that the proposed technique seems to be appropriated for compaction modeling.

REFERENCES

[1] Tekkaya A E, Schikorra M, Becker D, Biermann D, Hammer N, Pantke K. Hot profile extrusion of AA-6060 aluminum chips. J. of Mat Processing Technology 2009;209:3343–3350.

[2] Misiolek, W Z, Haase M, Khalifa N B, Tekkaya A E, Kleiner M. High Quality Extrudates from Aluminum Chips by New Billet Compaction and Deformation Routes. CIRP Annals - Manufacturing Technology 2012;61:239–242.

[3] Dragosek L, Kocisko R, Kovacova A, Bidulsky R, Skrobian M. The Compacting Process of The EN AW 6060 Alloy. Acta Polytechnica 2015;55(5): 301–305.

[4] Fleck N A, Kuhn L T, McMeeking R M. Yielding of Metal Powder Bonded by Isolated Contacts. J. of the Mech. and Phys. of Solids 1992;40(5):1139–1162.

[5] Frenning G. An Efficient Finite/Discrete Element Procedure for Simulating Compression of 3D Particle Assemblies. Comput. Methods Appl. Mech. Eng 2008;197:4266–4272.

[6] Jerier J K, Hathong B, Richefeu V, Chayreyre B, Imbault D, Donze F.V, Doremus P. Study of Cold Powder Compaction by Using the Discrete Element Method. Powder Technology 2011;208:537–541.

[7] Gustafsson G, Häggblad H, Jonsén P. Multi-particle Finite Element Modelling of the Compression of Iron Ore Pellets with Statistically Distributed Geometric and Material Data. Powder Technology 2013;239:231–238.

[8] Shima S, Oyane M. Plasticity Theory for Porous Metals. I. J. of Mechanical Science 1976;18:285-291.

[9] Doraivelu S. M, Gegel H. L, Gunasekera J S, Malas J C, Morgan J T. A New Yield Function for Compressible P/M Materials. I. J. of Mechanical Science 1984;26(9/10):527-535.

[10] Miller R. A Continuum Plasticity Model for the Constitutive and Indentation Behavior of Foamed Metal. I. J. of Mechanical Science 2000;42(4):729-754.

[11] Alves L, Martins P, Rodrigues J. A New Yield Function for Porous Materials. J. of Materials Processing Technology 2006;179:36–43.

[12] Han L H, Elliott J A, Bentham A C, Mills A, Amidon G E, Hancock B C. A Modified Drucker – Prager Cap Model for Die Compaction Simulation of Pharmaceutical Powders. I. J. of Solid and Structures 2008;45:3088-3106.

[13] Ramírez J F, Cardona M, Velez J A, Mariaka I, Isaza J A, Mendoza E, Betancourt S and Fernández-Morales P. Numerical Modeling and Simulation of Uniaxial Compression of Aluminum Foams using FEM and 3D-CT Images. Procedia Materials Science 2014;4:227-231.

[14] Alessia M, Giovanni M, Calogero C, D'Anna J, La Mendola L. Finite element analysis of the out-of-plane behavior of FRP strengthened masonry panels. Composites Part B: Engineering 2017; 115: 188-202.

[15] Girum S. Urgessa n, Arciszewski T. Blast response comparison of multiple steel frame connections. Finite Elements in Analysis and Design 2011;47:668–675.