Journal of Manufacturing Processes 20 (2015) 374–379

Contents lists available at ScienceDirect

Journal of Manufacturing Processes

journal homepage: www.elsevier.com/locate/manpro

Review

Dynamic modeling and simulation of the screwing behavior of thread forming screws

G. Dinger*

University of Siegen, Department of Mechanical Engineering, Institute of Engineering Design MVP, Paul-Bonatz-Str. 9-11, D 57076 Siegen, Germany

ARTICLE INFO

Article history:
Received 15 October 2014
Received in revised form 18 June 2015
Accepted 18 June 2015
Available online 9 July 2015

Keywords:
3D finite element analysis
Hole size
Material flow
EN AW7075
Fastening element
Thread rolling screw

ABSTRACT

The thread forming screw assembly process is evaluated with numerical simulation and experimental approaches. The paper presents a 3D finite element analysis of a thread forming process using the ABAQUS/Explicit software program. The aim of this work is to provide an improved understanding of the forming mechanism and to reduce experimental testing due to the use of the finite element method.

A parametrical study was also conducted to identify the influence of the friction and the hole diameter on the screwing torque. The study shows that the lead hole diameter has an important influence on the screwing torque. For the verification the numerical simulation results are compared with experimental results.

Contents

1. Introduction

Bolted joints are widely used in mechanical products and structures due to their possibility for easy disassembly and their relatively low cost [1]. Thread forming screws shape themselves without cutting during installation her threads in pre-drilled, punched, cored and extruded holes and eliminate the separate tapping operation which is necessary with traditional screws. Moreover thread forming screws avoided the chip evacuation problem. Although thread forming screws are frequently used assembly procedures in the automotive industry, the modeling and simulation of this phenomenon is by no means trivial. It is particularly complex due to the diversity of physical phenomena involved, including large elastic–plastic deformation, thermo-mechanical coupling and contact with friction. Thread forming screws are particularly used for the assembly of non-ferrous materials with highly ductile behavior.

However, only few studies related to 3D finite element analysis for the thread forming process have been published. Guan et al. [2] studies the stress distribution during a dynamic simulation of the dental implant insertion process. Mathurin et al. [3] determine displacements during the forming process and the shape of the threads with a 3D finite element model from a 45 degree sector (discrete rigid screw body). They also conducted a parametrical study to identify the most influential process parameter. They show that

* Current address: SIEGENIA-AUBI KG, Industriestraße 1-3, D 57234 Wilnsdorf, Germany. Tel.: +49 2713931 950.
E-mail address: contact@georg-dinger.de

http://dx.doi.org/10.1016/j.jmapro.2015.06.012
1526-6125/

the lead hole diameter has an important influence on the screwing torque.

Warrington et al. [4] investigate with a 3D model of a thread the split crests formation during tapping. They validated their model with linear scratch experiments and to actual tapping. Boicea et al. [5] have developed a numerical model for the analysis of the cold plastic deformation process with planetary rollers of profiles.

A bibliographical review of finite element methods for the analysis of fastening and joining is given by Mackerle [6].

This paper studies the forming mechanism and proves that a quantified prediction of the assembly process of the thread forming screws with numerical simulations is possible. Normally for every modification such assembly graph must be determined experimentally, which is time and cost intensive. In this work a thread forming screw according to DIN 7500 M8x35 with ISO metric threads, lens head according to DIN EN ISO 14583 is considered. The uniform trilobular cross section of the thread shape reduces the assembly torque during thread forming. A parametrical study is also conducted to identify the influence of the friction and the hole diameter on the screwing torque. For the verification the numerical simulation results are compared with experimental results.

2. Simulation

In the aim of developing a general simulation method for thread forming screws all important influences for the forming mechanism have to be considered in the numerical simulation. A three-dimensional finite element (FE) model is established.

2.1. Model description

In the numerical simulation the challenging problems are the large deformations in the cutting zone with can lead to element distortion, the several revolutions, which increase analysis time and the numerous elements, which increase the computational load.

The general purpose finite element analysis software Abaqus/Explicit 6.11.1 has been used. The simulation model is a replica of the experimentally parts and the assembly parameters used where the same in both processes. The heat is generated due the dissipation of plastic deformation and friction, being transferred through the parts and lost to the surrounding environment by convection and radiation. Due to the high assembly speed adiabatic conditions, where no heat transfer takes place, are assumed for the simulation. Application of the explicit dynamics to model quasi-static events such as the metal forming process in its natural time period is computationally impractical. Literally millions of time increments are needed. Therefore the mass scaling approach has been used to artificially increase the speed of the process in the simulation to obtain an economical solution.

Contact interactions must be enforced between the screw and clamped part and nut thread component. Friction along the screw-nut thread component contact interface, during the assembly process, is a very complex phenomenon and influences the thread geometry, temperature and wear. In this study the simple Coulomb friction model, is assumed for the given interfaces. The friction coefficient $\mu = 0.04$ has been measured with a fastener test bed (Fig. 7). In this study the general contact algorithm with a finite-sliding contact enforces contact interactions between all bodies in the model. The contact constraints are enforced with the penalty method. The experience of contact modeling for self-loosening simulation was considered for the simulation of the forming process [7]. The assumptions regarding heat transfer and friction modeling are also used for numerical metal cutting process simulations.

The lower part of the assembly has a pre-drilled hole. The thread has a uniform trilobular cross section. To facilitate insertion of the screw when assembling the end of the screw is formed with two and a half tapered threads. The complex geometry of the thread forming screw was generated with SolidWorks and imported and meshed with Abaqus/CAE 6.11.1.

Table 1
Coefficients of the Johnson–Cook model [8].

Material	A (MPa)	B (MPa)	n	C	m	T_{melt} (°C)
EN AW 7075	496	310	0.3	0	1.2	635

The nut thread component with blind hole is restrained in the x, y and z directions and the x and z directions of the clamped part are restrained at the surfaces marked in Fig. 1. The axial stiffness of the fastener test bed is considered with a spring and the axial force F_A is ramped linearly down over the step time.

In this study, EN AW 7075 is modeled with the constitutive plasticity model by Johnson-Cook. The flow stress is a function of the plastic strain ε, the strain rate $\dot{\varepsilon}$ normalized by the reference strain rate $\dot{\varepsilon}_0$ and the homologous temperature T^*, as shown in Eq. (1).

$$\sigma = (A + B\varepsilon^n)\left(1 + C\ln\left(\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right)\right)(1 - (T^*)^m) \tag{1}$$

where A, B, C, m, n are material constants. Coefficients of law are given in Table 1.

Elastic material behavior was assumed for the screw steel, with a young's modulus and Poisson's ratio of 210 GPa and 0.29, respectively. The properties of screw, clamped part and nut thread component are mentioned in Table 2.

The three main model formulations are the Lagrangian, Eulerian and the arbitrary Lagrangian–Eulerin (ALE). The traditional Lagrangian formulation is used, where the finite element mesh is attached to the material and deform together with the material during the assembly process. This formulation generally offers computational efficiency and accuracy, but tend to exhibit mesh distortion and an associated loss of accuracy when undergoing large deformations. Although large deformations are expected, no special solution controls or analysis techniques (e.g. adaptive meshing) are applied to the model.

Reduced integrated hexahedral eight node elements (C3D8R) were used for all parts and 72 linear wedge elements (C3D6) in the hole starting zone of the nut thread component. To improve the accuracy of the solution and to avoid invalid element distortions due to large deformations that are generated during the thread forming process, the mesh of the nut thread component has been optimized. The mesh is helix shaped with the same pitch as the screw and hourglass and distortion control is used. Due to the large number of revolutions of the screw the second-order accuracy formulation is used for the solid screw elements. For all other parts the default first-order accuracy formulation is used. The nut thread component is meshed with 60 elements over the circumference (Fig. 2).

The mesh density of the screw was checked by convergence tests and comparison with analytical calculations of the axial stiffness of the fastener system according to VDI 2230 [9]. The whole assembly is meshed with 167,900 elements. The energy balance has been checked to validate the energy conservation in the numerical model during the assembly process.

Table 2
Thermo mechanical properties of screw, clamped part and nut thread component.

Physical parameters	Steel	EN AW 7075
Density ρ (g cm^{-3})	7.83	2.7
Elastic modulus E (GPa)	210	73.1
Poisson's ratio ν (–)	0.29	0.33
Specific heat C_p (J g^{-1} °C^{-1})	0.475	0.875
Thermal conductivity λ (W m^{-1} °C^{-1})	44.5	121
Expansion α (μm m^{-1} °C^{-1})	13.2	23.2
Inelastic heat fraction	–	0.9