Send Orders for Reprints to reprints@benthamscience.ae

Open Access

The Nonlinear Dynamic Behavior Analysis and Research for Sheet Steel Stamping Forming Problems Based on Finite Element Method

Li Zhang[1,2], Tao Zhang[*,2] and Kaiteng Wu[2]

[1]State Key Laboratory of Explosion Science and Technology, Beijing Institute of Technology, Beijing 100081, China

[2]Key Laboratory of Numerical Simulation of Sichuan Province, Neijiang Normal University, Neijiang 641110, China

Abstract: A numerical framework for the simulation of sheet steel stamping forming is presented. The main problems, the equations of motion, the constitutive relation, the initial conditions, boundary conditions and contact conditions, are presented in detail. Based on this, the finite element model is established and solved for exploring the changes in laws of stress, strain and so on. The information on stress, strain and load displacement is obtained at different deformation stages. The numerical results show that the finite element algorithm can effectively simulate the deformation process of sheet steel which helps to explain that the numerical framework is feasible for sheet steel stamping forming problems.

Keywords: Finite element algorithm, nonlinear dynamic behavior, sheet steel, stamping forming.

1. INTRODUCTION

The metal forming problems are advantageous for the characteristics of metal plastic deformation. By adding pressure on the metal through a certain way, the metal produces plastic deformation, which is required for various shapes [1]. Metal stamping molding technology has a long history. For example, it was used to make metal vessels and COINS by the people of the pre-historic era. With the development of computer technology and finite element method, and in order to make the stamping forming process design and calculation more accurate, people began to study numerical analysis means of forming process for all kinds of physical phenomena. The metal forming has a very wide application in machinery, automobile, aerospace, national defense war industry, household appliances, packaging and other fields. It is very important in modern manufacturing technology of material processing technique [2]. The metal forming process involves geometric nonlinearity, material nonlinearity and boundary conditions of complex mechanical problems. In general, stamping forming process contains four important physical phenomena, namely contact collision phenomenon, friction and wear phenomenon, large displacement, large rotation and deformation phenomenon, the elastic-plastic deformation phenomenon. For these reasons, it is very difficult to control stamping forming process, and there is a need to go through trial and error method to produce appropriate products. This process could take a lot of time and money. Based on the theory of the finite element, numerical simulation method is gradually applied in stamping die design and process optimization. Thus, it greatly reduces the mold test repeatedly and optimizes the experimental process, saves a large amount of manpower and material resources, reduces the production cycle and saves the cost of production. A large number of domestic and foreign scholars have done a great job of theory, experiment and simulation about metal stamping forming. For example, sheet metal forming simulation has been studied by finite element and Monte Carlo algorithm [3, 4]. The analysis of spring back in sheet metal bending has been researched using finite element method [5]. An incremental and implicitly iterative finite element method combined with orthogonal regression has been researched for stamping process design [6].

In this paper, as the research object is a sheet steel, the dynamic behavior in the forming process is discussed by finite element method. This paper aims to present a numerical framework for the simulation of sheet steel stamping forming. It shows that numerical simulation results fit well with the actual situation.

2. MECHANICAL MODEL

Sheet stamping forming process is a typical contact collision problem. There are only two body contact systems [7] (see Fig. **1**). In Fig. (**1**), ${}^{t}x_1, {}^{t}x_2, {}^{t}x_3$ stand for spatial location fixed orthogonal coordinate system, ${}^{0}\Omega^1, {}^{0}\Omega^2$ stand for object 1 and 2 of the space in a reference time $t=0$, ${}^{t}\Omega^1, {}^{t}\Omega^2$ stand for object 1 and 2 of the space in a reference time *t*. The purpose of discussing the system is to study the response for a given contact system from the reference point $t=0$ to the given moment $T>0$.

For stamping forming process, the time duration required is the period of time for sheet metal from the start of molding until the pressure is completely removed. A contact system response mainly has four categories of equation constraints, which are motion equation, constitutive equations, boundary conditions and initial conditions. The details are discussed below.

*Address correspondence to this author at the Key Laboratory of Numerical Simulation of Sichuan Province, Neijiang Normal University, Neijiang 641110, China; Tel: +8613696066898; Fax: +8608322341679;
E-mail: taozhangnjtc@126.com

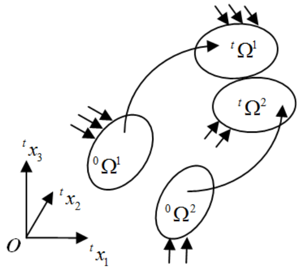

Fig. (1). Two-body contact system schematic diagram.

2.1. Equations of Motion

$$\frac{{}^t\sigma_{ji}}{x_j} + {}^t b_i = {}^t\rho {}^t a_i \; (i=1,2,3 \; j=1,2,3) \tag{1}$$

where σ_{ji} is the Cauchy stress, ${}^t b_i$ is the volume force component, ${}^t a_i$ is the acceleration component.

2.2. Constitutive Relation

Different materials have different constitutive relations; even the same kind of material, under different deformation condition may have a different constitutive relation. Describing online material deformation within the elastic range, the generalized Hooke's law can be used.

$${}^t S_{ij} = C_{ijk1} {}^t\varepsilon_{k1} \; (i,j,k=1,2,3) \tag{2}$$

where ${}^t S_{ij}$ denotes stress tensor, ε_{k1} denotes strain tensor, and C_{ijk1} denotes material constant.

Because strain tensor and strain gage are symmetric, vector can be introduced to represent the stress and strain, namely

$${}^t\sigma = ({}^t\sigma_{11}, {}^t\sigma_{22}, {}^t\sigma_{33}, {}^t\sigma_{12}, {}^t\sigma_{23}, {}^t\sigma_{31})^T \tag{3}$$

$${}^t e = ({}^t e_{11}, {}^t e_{22}, {}^t e_{33}, {}^t e_{12}, {}^t e_{23}, {}^t e_{31})^T \tag{4}$$

So, Hooke's law can be written in the following form

$${}^t\sigma = C {}^t e \tag{5}$$

where C is a material constant matrix (6×6). For isotropic elastic material, there is

$$C = \begin{bmatrix} \lambda+2\mu & \lambda & \lambda & 0 & 0 & 0 \\ \lambda & \lambda+2\mu & \lambda & 0 & 0 & 0 \\ \lambda & \lambda & \lambda+2\mu & 0 & 0 & 0 \\ 0 & 0 & 0 & \mu & 0 & 0 \\ 0 & 0 & 0 & 0 & \mu & 0 \\ 0 & 0 & 0 & 0 & 0 & \mu \end{bmatrix} \tag{6}$$

where λ and μ are the elastic constants. They can be expressed as following

$$\lambda = \frac{vE}{(1+v)(1-2v)} \tag{7}$$

$$\mu = G = \frac{E}{2(1+v)} \tag{8}$$

where G is the shear modulus, E is the Young's modulus, and v is the Poisson's ratio.

Then type (6) can become:

$$C = \begin{bmatrix} 1-v & v & v & 0 & 0 & 0 \\ v & 1-v & v & 0 & 0 & 0 \\ v & v & 1-v & 0 & 0 & 0 \\ 0 & 0 & 0 & 1-2v/2 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1-2v/2 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1-2v/2 \end{bmatrix} \tag{9}$$

Sheet forming process will produce plastic deformation. Therefore, it is necessary to establish the elastoplastic constitutive relation. As an empirical fact, when the stress satisfies the following conditions (10), named as yield criterion, material plastic begins to deform.

$$\Phi(\sigma,\kappa) = 0 \tag{10}$$

where σ is the stress tensor, and κ is an internal variable depending on the plastic deformation.

In general, the following guidelines determine the material yield.

If $\Phi < 0$, $(\frac{\Phi}{\sigma ij})\sigma ij < 0$ or $\Phi = 0$, $(\frac{\Phi}{\sigma ij})\sigma ij < 0$, it is elastic deformation.

If $\Phi = 0$, $(\frac{\Phi}{\sigma ij})\sigma ij \geq 0$, it is plastic deformation.

Von Mises yield criterion is a more general yield criterion, which is particularly applicable to metal materials. The equivalent stress of Von Mises yield criterion is as follows.

$$\sigma_e = \sqrt{\frac{1}{2}[(\sigma_1-\sigma_2)^2 + (\sigma_2-\sigma_3)^2 + (\sigma_1-\sigma_3)]} \tag{11}$$

where σ_1, σ_2, σ_3 are three principal stress.

2.3. Initial Conditions

Assuming that the initial contact body stress is zero, the initial displacement and velocity are defined as,

$$u(X,0) = {}^0 u \tag{12}$$

$$v(X,0) = {}^0 v \tag{13}$$

where ${}^0 u$ expresses the initial displacement, and ${}^0 v$ expresses the initial velocity.

2.4. Boundary Conditions

Boundary conditions usually include the given boundary displacement and the given boundary.

$$u(X,t) = {}^t u, \text{on the border } {}^0\Gamma_D \tag{14}$$

$$ {}^t\sigma_{ij}\,{}^tN_{1j} = {}^t q, \text{ on the border } {}^0\Gamma_F \tag{15}$$

where ${}^tN_{1j}$ is the jth component of unit normal vector ${}^tN_1$ on the border ${}^0\Gamma_F$.

2.5. Contact Conditions (On the Contact Boundary)

$$ p = ({}^tV_1 - {}^tV_2)\,{}^tN_1^1 \le 0 \tag{16}$$

$$ {}^tq_1 \le 0 \tag{17}$$

$$ {}^tq_1 = [({}^tq_2)^2 + ({}^tq_3)^2]^{1/2} \le v^t q_1 \tag{18}$$

$$ {}^tV = 0,\ {}^tq_t < v^t q_1 \tag{19}$$

$$ {}^tV = -\lambda\,{}^tq_t^2,\ {}^tq_t = v^t q_1 \tag{20}$$

where ${}^tV_1, {}^tV_2$ are the velocity, ${}^tq_i (i = 1,2,3)$ is the direction of the friction in the *x*, *y*, *z* directions. Friction meets the coulomb friction law.

3. THE NUMERICAL SIMULATION STEPS OF THE FINITE ELEMENT METHOD

For the stamping forming process, the general process is described as follows:

(i) The mechanical model should be set up.

(ii) Finite element analysis model based on mechanics model should be set up.

(iii) Unit types should be selected and related parameters should be determined according to the sheet deformation characteristics.

(iv) Elastoplastic constitutive relation and its related parameters should be selected according to the sheet deformation characteristics.

(v) The law of friction and parameters should be selected according to the sheet metal features, die surface features and lubrication condition.

(vi) Solution.

(vii) Post processing.

4. AN ENGINEERING EXAMPLE

4.1. Model Description

It is assumed that the size of sheet steel is greater than the thickness of two directions, thus the problem can be simplified as plane strain problem. Since the bilateral of plane model are symmetrical, we need to consider half of the plane model, the mechanical model is shown in Fig. (**2**).

In Fig. (**2**), the length of sheet steel is *L*/2, the thickness of the sheet steel is *H*, and $L_2 = L_1 + H$. Table **1** shows the basic data of the model.

4.2. Select Unit Types and Material Constant

As stamping is a large deformation nonlinear problem, the four nodes plane unit PLANE42 has proved to be better through an experimental study of numerical simulation effect. Our simulation assumes that punch and die are the rigid material and the stress-strain relationships are the bilinear stress-strain, as shown in Fig. (**3**).

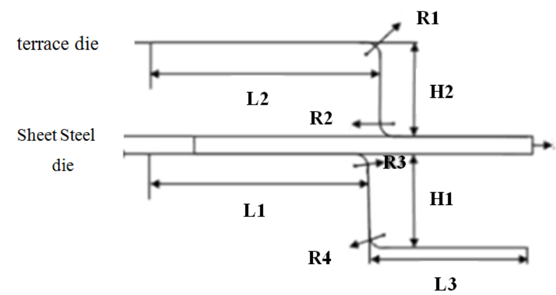

Fig. (2). Model sketches of sheet steel.

Table 1. The basic geometry model.

L	*H*	L_1	H_1	H_2
0.1m	0.004m	0.08m	0.02m	0.02m
L_3	R_1	R_2	R_3	R_4
0.04m	0.006m	0.006m	0.006m	0.006m

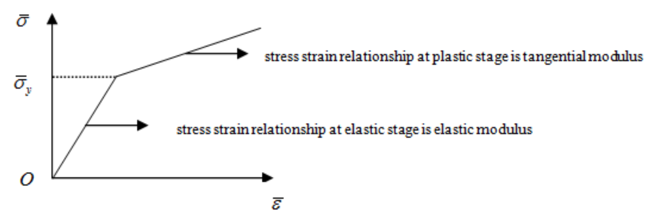

Fig. (3). Stress strain curve.

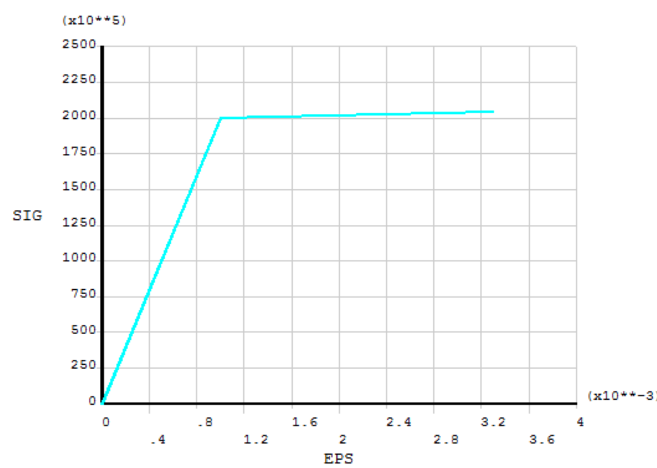

Fig. (4). Stress strain curve of sheet steel.

The young's modulus is 2E11pa, the poisson's ratio is 0.3, and the initial yield stress is 2E8pa. In order to simplify the problem, all frictions are 0.3. So, the relationships between stress and strain of steel sheet are shown in Fig. (**4**).

4.3. Boundary Conditions

Considering that the whole stamping process is static, all degrees of freedom for female die should be fixed, the translational and rotational degrees of freedom are restricted in the X orientation for male die, and the displacement ($-H_1$) in the Y orientation is forced for male die. It should be assumed that the right plane of the sheet steel is a symmetrical constraint. The displacement ($0.2\times H_2$) is again forced after stamping contact in the Y orientation.

4.4. Mesh Generation

In this paper, the finite element software ANSYS13.0 [8, 9] is used to simulate sheet steel stamping forming. For large deformation problem of sheet stamping process, we divide the sheet steel by mapping, the grid size is 0.002, and the shape of unit is Quad (see Fig. **5**).

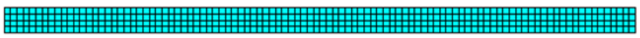

Fig. (5). Meshing figure.

4.5. Astringency

Stamping is a large deformation nonlinear problem. In order to guarantee the overall convergence of the algorithm, the Newton-Raphson method is used to solve the process. Fig. (**6**) shows the relationships between absolute convergence criterion and cumulative iteration numbers about the finite element algorithm, and it also explains the effectiveness of the algorithm.

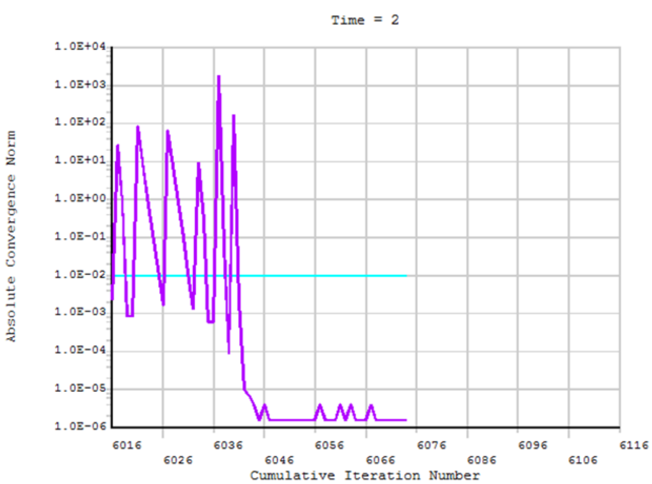

Fig. (6). The relationships between absolute convergence criterion and cumulative iteration numbers.

4.6. The Results of Numerical Simulation and Analysis

Figs. (**7**, **8**) respectively are the equivalent stress nephogram and the equivalent plastic strain nephogram of sheet steel. We can see from the pictures that both sides of sheet steel belong to the elastic strain without plastic strain. The stress is mainly distributed in the fillet. The residual stress is 284MPa and the location of residual stress is also in the fillet. In fact, in the stamping process, damage and micro-crack always appeared in the bending section easily. We must attach importance to it because the micro-crack should evolve into macroscopic cracks or fatigue life. On the other hand, the surface contact stress situation also shows the fatigue phenomenon. In Fig. (**9**), the maximum contact stress is still focused on the fillet. So, the simulation results correspond to the actual condition very well. And likewise, plastic strain has the similar results. From the data, we can know the stress is 606MPa after complete stamping. It states that the stress reduced about 300 MPa through rebound.

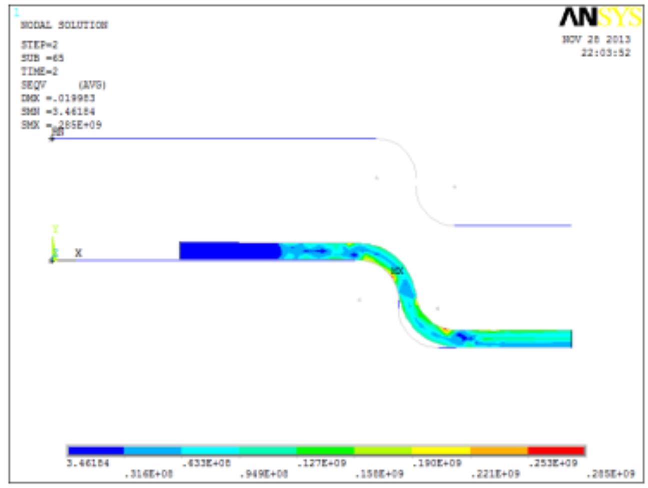

Fig. (7). The equivalent stress nephogram of sheet after spring back

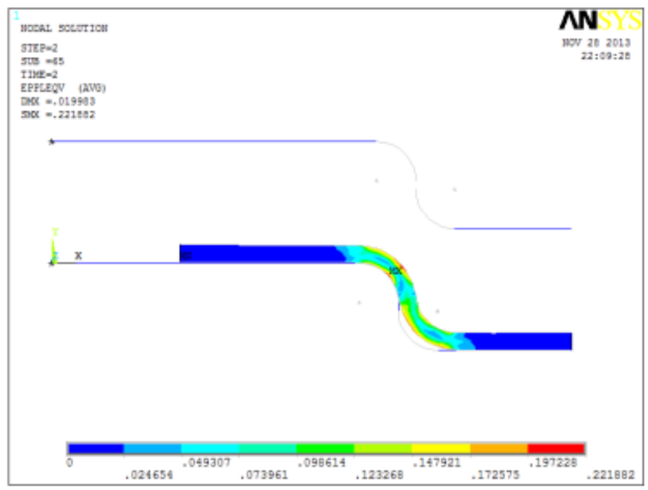

Fig. (8). The equivalent strain nephogram of sheet after spring back.

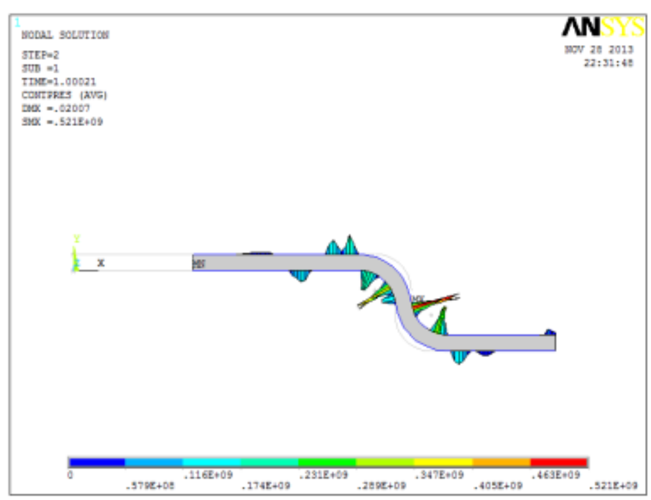

Fig. (9). Contact stress curve.

In general, when the structure has a large deformation, we should consider the load change. As you can see from Fig. (**10**), on the curve of load versus indentation depth, besides the straight parts correspond with the elastic property of sheet steel, the sudden drop of the load occurred several times. This phenomemon shows that sheet steel has a large deformation in the stamping process.

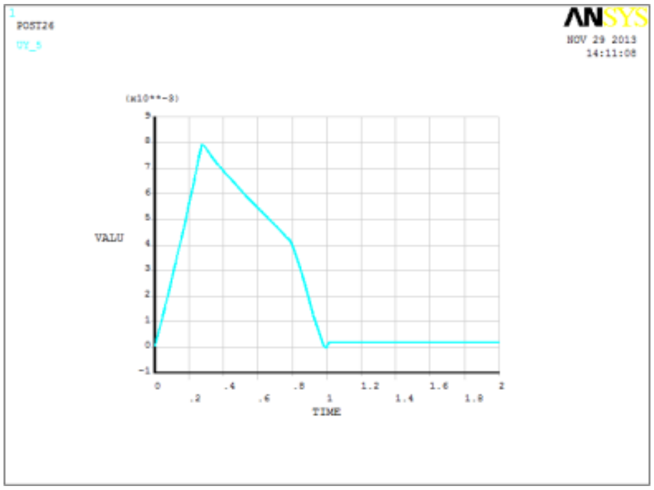

Fig. (10). The load-displacement.

Above all, the results show that the sheet steel in the adjacent area of die and punch deforms primarily. The maximal value of stress and strain has appeared in the outboard punch round corners. The maximum number of each output is presented in Table **2**, and these values can be referenced in the working out processing scheme of physical model.

Table 2. The maximum value of each output.

Stress	Strain	Counterforce
671.24	383.67×10^{-3}	457.15

CONCLUSION

In this paper, we have presented a numerical framework for the simulation of sheet steel stamping forming. This framework is based on finite element method. The modeling and solving steps are derived in the paper. By numerical simulation, the numerical simulation results fit well with the actual situation, which also confirms the practical effectiveness of this numerical framework. However, the whole stamping process as quasi-static without considering the influence of temperature, may be reduced to the precision of calculation to a certain degree.

CONFLICT OF INTEREST

The authors confirm that this article content has no conflict of interest.

ACKNOWLEDGEMENTS

This work was financially supported by the National Natural Science Foundation of China(10872085), the education department of Sichuan Province major project (12ZB266), the education department of Sichuan Province university innovation team(13TD0001), and the Neijiang normal university major achievements transformation projects (14CZ02).

REFERENCES

[1] Moriya T, Takamura M, Takahashi S, *et al.* Development of total stamping simulation system for evaluating formability of sheet metal forming parts. Rev Automot Eng 2009; 30(4): 449-56.

[2] Giuseppe I, Giuseppina A, Francesco G, Lorenzo D, Rosa A. A sustainability point of view on sheet metal forming operations: material wasting and energy consumption in incremental forming and stamping processes. J Mater Process Technol 2012; 29(3): 255-68.

[3] Yang Z, Lian-Hong Z, Shuang-Yi L, Hong-Zhi W. Sheet metal forming simulation and its application in stamping process of automobile panel. Eng Comput 2012; 8(4): 269-72.

[4] Mao-lin Y, Du R. Sensitivity analysis of the sheet metal stamping process based on inverse finite element modeling and Monte Carlo simulation. USA: AIP Conf Proc 2005.

[5] Panthi SK, Ramakrishnan N, Pathak KK, *et al*. An analysis of springback in sheet mental bending using finite element method (FEM). J Mater Process Technol 2007; 186(1-3): 120-4.

[6] Mehmet F, Osman H, Umit K, *et al*. Stamping process design FEA in conjunction with orthogonal regression. Finite Elem Anal Des 2010; 46(11): 992-1000.

[7] Guang-yao L, Zhi-hua Z. The computer simulation and application of sheet stamping forming process. Beijing: Beijing Institute of Technology Press 1998 (In Chinese).

[8] Hong-Cai Z, Bo H. Finite element analysis of ANSYS13.0 from entry to actual combat. Beijing: Mechanical Industry Press 2013 (In Chinese).

[9] Ze-Peng W, Ren-Xi H, Shi-Ting K. ANSYS13.0/LS-DYNA Nonlinear finite element analysis of examples of the tutorial. Beijing: Mechanical Industry Press 2011 (In Chinese).

Received: August 16, 2014 Revised: December 29, 2014 Accepted: January 7, 2015