Elastic and Plastic Mechanical Properties Determined by Nanoindentation and Numerical Simulation at Mesoscale

F. Y. Chen \(^1,a\) and R. C. Chang \(^2,b\)

\(^1\) Department of Mechanical Engineering, Lee-Ming Institute of Technology, Taiwan 24305
\(^2\) Department of Mechanical and Computer-Aided Engineering, St. John's University, Taiwan 25135
\(^a\) fychen@ms11.hinet.net, \(^b\) rcc@mail.sju.edu.tw

Keywords: nanoindentation; FEM; elastic-perfectly plastic; elastic-linear work-hardening.

Abstract. This work presents a comparison of numerical simulation and experiment of nanoindentation testing. A commercial finite element code ANSYS is adopted in the numerical simulation, in which elastic-plastic properties are considered. A PMMA specimen and a three side pyramidal Berkovich probe tip is used in the indentation tests. While the elastic-linear work-hardening properties are adopted, the numerical results agree well with the experimental data for different indentation loads. It proves the numerical simulation can be used in the small scale analysis.

Nanoindentation Test

In experiments, a nanoindenter (Triboscope, Hysitron) assembled on an atomic force microscope (Autoprobe, CP-Research) is adopted, where the specimen is PMMA and a three side pyramidal Berkovich probe tip is used in the test. The nanoindentation testing is successfully proved to evaluate the hardness and the elastic modulus of materials. Nanoindentation is performed under a precisely continuous measurement of the force (load) and the displacement (depth) during the test. Fig. 1 shows a schematic draw of an indentation load via depth curve.

Fig. 1 Loading and unloading curve.

Using the Oliver and Pharr [1] method, the elastic modulus of the indented sample can be inferred from the initial unloading contact stiffness, \( S = dp / dh \), i.e., the slope of the initial portion of the unloading curve, as shown in Fig. 1. Based on the relationships developed by Snedden [2] for the indentation of an elastic half space by any punch that can be described as a solid of revolution of a smooth function, a geometry-independent relation involving contact stiffness, contact area, and elastic modulus can be derived as follows

\[
S = 2 \beta \sqrt{\frac{A}{\pi}} E_r, \tag{1}
\]

where \( \beta \) is a constant that depends on the geometry of the indenter, for Berkovich indenter
\[ \beta = 1.034 \] and \( E_s \) is the reduced elastic modulus, which accounts for the fact that elastic deformation occurs in both the sample and the indenter. \( E_i \) is given by

\[ E_i = \frac{1 - \nu^2}{E} + \frac{1 - \nu_i^2}{E_i}, \]  

(2)

where \( E \) and \( \nu \) are the elastic modulus and Poisson’s ratio for the sample, respectively, and \( E_i \), \( \nu_i \) are the same quantities for the indenter. For an indenter with a known geometry, the projected contact area is a function of the contact depth. The area function for a perfect Berkovich indenter is given by

\[ A = f(h_c) = 24.56h_c^3. \]  

(3)

Indenters used in practical nanoindentation testing are not ideally sharp. Therefore, tip geometry calibration or area function calibration is needed. A series of indentations is made on fused quartz at depths of interest. A plot of \( A \) versus \( h_c \) can be curve fit according to the following functional form

\[ A = f(h_c) = 24.56h_c^3 + C_1h_c^4 + C_2h_c^{1/2} + C_3h_c^{1/4} + \ldots + C_8h_c^{1/128}, \]  

(4)

where \( C_1 \) through \( C_8 \) are constants. The lead term describes a perfect Berkovich indenter.

**Numerical Simulation**

The FEM has been widely used to simulate the nanoindentation process [3-6]. In this study, a commercial finite element code ANSYS is used. The linear elastic, elastic-perfectly plastic, and elastic-linear work-hardening materials are considered. The linear elastic modulus \( E \) is calculated by experiment data from equations (1)-(4) while Poisson’s ratio is defined as \( \nu = 0.3 \). In 2D simulation, the Plan42, Cona171 and Targe169 elements of ANSYS are adopted. Fig. 2 shows the 2D FEM mesh, using an axisymmetric cone with half-included angle of 70.3° in which the conical indenter has the same area function as a Berkovich tip [3]. In 3D simulation, the Solid95, Conta174 and Targe170 elements are employed. Fig. 3 shows the 3D FEM mesh where a three sided pyramidal Berkovich probe tip with angle of 142.3° is used.

![Fig. 2 2D FEM mesh](image1)

![Fig. 3 3D FEM mesh](image2)

**Results and Discussion**

The nanoindentation is conducted by using a load-time sequence as shown in Fig. 4. The indenter is first loaded and unloaded three times in succession at a constant rate to ensure that contact is maintained between the specimen and the indenter, where the unloading terminated at 10% of the peak load. Then, the loading, holding, and unloading are performed. All the data can be found from the resulting load-depth curve. The load-depth curves of different loading are shown in Fig. 5. From those load-depth curves, the corresponding elastic modulus of different loading can be calculated by using Eqs. (1) to (4) with \( E = 6.165 \text{GPa} \) and \( P = 935 \mu \text{N} \).
Before the numerical simulation, convergence studies of the finite element method must be done. First, the material properties are assumed as linear elastic, the material properties of the specimen are $\nu=0.3$, $E=6.165\text{GPa}$, and $\nu=0.07$, $E=1140\text{GPa}$ for the indenter. The convergence studies of the finite element method of the different element sizes by using 2D model (Fig. 2) and 3D model (Fig. 3) are calculated. Fig. 6 illustrates the comparison of the load-displacement curves of the experiment and the numerical simulation for the force $P=935\mu\text{N}$. The element size is smaller than 400nm in the 2D case and element number is larger than 86900 in the 3D case. The numerical results of the 2D and the 3D simulation are very close, but they are much smaller than the experimental data.

Second, the material properties are assumed to be elastic-perfectly plastic (Fig. 7). Fig. 8 presents the load-depth curves in different yield stress $S_y$, where the linear elastic modulus is $E=6.165\text{GPa}$ and force is $P=935\mu\text{N}$. The figure shows, while the load is smaller than 750$\mu\text{N}$, all of the curves by FEM are very close to the linear elastic one (dashed line), but are far away from the experimental data.

**Fig. 4** Load sequence. **Fig. 5** Load-depth curves in different loading

**Fig. 6** Load-depth curves by different methods. **Fig. 7** Elastic- perfectly plastic stress-strain curve

**Fig. 8** Load-depth curves by different yield stress. **Fig. 9** Elastic- elastic-linear work-hardening stress strain curve
Finally, the material properties are assumed to be elastic-linear work-hardening as showed in Fig. 9, where $E_2$ denotes the work-hardening rate. Fig. 10 shows the load-depth curves in different work-hardening rate $E_2$ at the yield stress $\sigma_y=0.01GPa$, where the linear elastic modulus is $E=6.165$ GPa, and force is $P=935\mu N$. From Fig. 10, the numerical results agree well with the case that the work-hardening rate is nearly $E_2=2.5Gpa$. Fig. 11 reveals the effect of various yielding stress $\sigma_y$ while the work-hardening rate is $E_2=2.5GPa$. The figure shows that the load-depth curves of FEM are more close to the experimental curves at the yield stress $\sigma_y=0.01GPa$. According to the above discussion, the load-displacement curves of numerical simulation agree well with the experimental data when the elastic-linear work-hardening properties are adopted in ANSYS.

Conclusions

The response of nanoindentation is simulated by FEM while the 2D and 3D models are considered. The numerical results are compared with the experiment data, and the following conclusions are obtained.

(1) The numerical simulation can be used in the small scale analysis. Both the 2D and 3D FEM models are convergent well. The 2D FEM model can be used in the simulation of a three sided pyramidal Berkovich probe tip if the conical indenter has the same area function as the Berkovich tip.

(2) While the elastic-linear work-hardening properties $E_2=2.5Gpa$ and $\sigma_y=0.01Gpa$ are adopted in ANSYS, the load-displacement curves of numerical simulation are very close to the experimental curves.

REFERENCES