Numerical simulation of transient temperature and residual stresses in friction stir welding of 304L stainless steel

X.K. Zhu, Y.J. Chao*

Department of Mechanical Engineering, University of South Carolina, Columbia, SC 29208, USA

Received 21 February 2002; accepted 30 October 2003

Abstract

Three-dimensional nonlinear thermal and thermo-mechanical numerical simulations are conducted for the friction stir welding (FSW) of 304L stainless steel. The finite element analysis code—WELDSIM, developed by the authors specifically for welding simulation, was used. Two welding cases with tool rotational speeds of 300 and 500 rpm are analyzed. The objective is to study the variation of transient temperature and residual stress in a friction stir welded plate of 304L stainless steel.

Based on the experimental records of transient temperature at several specific locations during the friction stir welding process for the 304L stainless steel, an inverse analysis method for thermal numerical simulation is developed. After the transient temperature field is determined, the residual stresses in the welded plate are then calculated using a three-dimensional elastic–plastic thermo-mechanical simulation. The effect of fixture release after the welding on the residual stresses is also studied. Comparison with the residual stress fields measured by the neutron diffraction technique shows that the results from the present numerical simulation have good agreement with the test data.

Keywords: Friction stir welding; Finite element analysis; Transient temperature; Residual stress; Numerical simulation; 304L stainless steel

1. Introduction

Friction stir welding (FSW) is a relatively new, state-of-the-art solid state joining process. This metal jointing technique is derived from the conventional friction welding. In a typical FSW, a rotating cylindrical pin tool is forced to plunge into the plates to be welded (i.e. workpiece) and moved along their contact line. During this operation, frictional heat that is generated by contact friction between the tool and workpiece softens the material. The plasticized material is stirred by the tool and forced to “flow” to the side and the back of the tool as the tool advances. As the temperature cools down, a solid continuous joint between the two plates is then formed. Because the highest temperature in the FSW process is lower than the melting temperature of the workpiece material, FSW yields fine microstructures, absence of cracking, low residual distortion and no loss of alloying elements that are the main advantages of this solid phase process. Nevertheless, as in the traditional fusion welds, a softening heat affected zone and a tensile residual stress parallel to the weld also exist.

Although it is a new welding technology, the FSW has been extensively studied in both the academic and industrial communities for most aluminum alloys including difficult-to-weld alloys such as AA2195 (with lithium) and AA7075. To date, most of the researchers focused their attentions on the heat transfer or temperature analysis of FSW. Tang et al. [1] presented the experimentally measured temperature distributions of the workpiece in an FSW. Gould and Feng [2] proposed a simple heat transfer model for predicting the temperature distribution in the workpiece of the FSW. Chao and Qi [3,4] developed a moving heat source model in a finite element analysis and simulated the transient temperature, residual stress and residual distortion of the FSW process. Furthermore, Colegrove et al. [5] and Frigaard et al. [6] developed three-dimensional heat flow models for the prediction of temperature fields in the FSW. Midling [7], Russell and Sheercliff [8] investigated the effect of tool shoulder material and pin tool on heat input during the FSW. Most currently, Donne et al. [9] reported the measured residual stresses in friction stir welds for 2024-T3 and 6013-T6 aluminums. Dong et al. [10] carried out a coupled thermo-mechanical analysis of the FSW process using a simplified two-dimensional axisymmetric model. Chao et al. [11] investigated the variations of heat...
energy and temperature produced by the FSW in both the workpiece and the pin tool. All investigations show that the FSW process of aluminum alloys yield welds with low distortion, high quality and low cost. Consequently, better structural performance is the primary advantage of this technology’s applications. For example, a demonstration of the tremendous potential and successful applications of aluminum FSW in airframe structures can be found in Talwar et al. [12].

In principle, the FSW process can be applied to joining other alloy materials such as steels and titanium as well. Of course, it is well known that current tool materials used in the FSW for aluminum are not adequate for production applications in many of the harder alloy materials. However, when adequate wear resistant tool materials become available, the benefits of the FSW may promote its rapid implementation in the production of ferrous structures and structures made from other more refractory materials. While we work to develop the necessary tool materials continues, it is also important to make progress in the development of the FSW process for steels. For instance, experimental studies of austenitic stainless steels [13] revealed the microstructures, residual stresses and strength of the friction stir welds. To further understand the fundamental mechanisms associated with the welding formation process and improve the welding quality for the FSW of steels, numerical modeling and simulations of transient temperature and residual stresses are valuable and necessarily needed.

For this purpose, the present paper conducts three-dimensional nonlinear thermal and thermo-mechanical simulations for the FSW of 304L stainless steel using the finite element analysis. Two welding cases with tool rotational speeds of 300 and 500 rpm are considered, respectively. Based on the experimental data of transient temperature history at specific locations during FSW, an inverse analysis method for the thermal numerical simulation is first developed. The transient temperature and residual stresses with and without the fixture release on the workpiece is subsequently determined numerically. The results for both transient temperature and residual stresses are compared with the available experimental data to validate the present simulations.

2. Geometry configuration and computational formulation

2.1. Geometry configuration

Two thin plates of 304L stainless steel were welded together using the FSW process. The pin tools were made of tungsten alloys in relative simple geometry. The welding was performed on the friction stir welding process development system, FSW-PDS (manufactured by the MTS Systems Corp.) at the University of South Carolina. Each plate has 304.8 mm (12 in.) in length, 101.6 mm (4 in.) in width and 3.18 mm (1/8 in.) in thickness. The pin tool has a shoulder diameter of 19.05 mm (3/4 in.), and a pin diameter of 6.35 mm (1/4 in.). The geometry of workpiece and pin tool in a typical FSW is shown in Fig. 1. The plates are supported on a large base plate, and fixed through clamping by a 304.8 mm long L-shaped steel strip (25.4 mm × 25.4 mm × 6.35 mm) on each plate at the distance 50.8 mm (2 in.) from the weld centerline. After the weld is cool down to the room temperature, the fixture is released. The pin tool starts at 6.4 mm (1/4 in.) away from the edge, and stops after translation of 279.4 mm (11 in.) along the weld line. Two welds were made independently using tool rotational speeds of 300 and 500 rpm, respectively. In both cases, the plunging force (or the Z force) is 31,138 N (7000 lb); the tool translated with a velocity of 1.693 mm/s (4 in./min); and the total welding time is 165 s for each weld.

The welds of the 304L plates were made in a single pass. The temperature history is recorded during the FSW process by 36 gauge K type thermocouples at nine locations on the top and bottom surfaces along the transverse sections near the middle of the plate. The locations of these points in the workpiece for the case having rotational speed of 300 rpm are shown in Fig. 2. The top row has five thermocouples located at 15, 18, 21, 23.5 and 26.5 mm, respectively. The bottom row is on the back of the plate, and has four thermocouples located at 12.7, 18, 21, and 27.5 mm, respectively. The locations of the thermocouples for the 500 rpm case are only slightly different from those shown in Fig. 2.

2.2. Inverse analysis method for heat transfer

An uncoupled thermal and thermo-mechanical analysis is adapted in this work, which is similar to the numerical simulation of the conventional arc welding [14,15]. The heat transfer analysis was performed first, and the transient temperature outputs from this analysis are saved for the subsequent thermo-mechanical analysis. In the thermal analysis, the transient temperature field T is a function of time t and the spatial coordinates (x, y, z), and is determined by the three-dimensional nonlinear heat transfer equation [16]:

\[ \dot{\kappa} \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) + Q_{int} = \rho c \frac{\partial T}{\partial t} \]  

(1)

where \( \kappa \) is the coefficient of thermal conductivity, \( Q_{int} \) is the internal heat source rate, \( \rho \) is the mass-specific heat capacity.

\[ \begin{array}{ccc}
203.2 \text{ mm} & 3.18 \text{ mm} \\
304.8 \text{ mm} \\
\end{array} \]

![Fig. 1. Geometry configuration of friction stir welding.](image-url)
and ρ is the density of materials. Heat flux to the system is put in by a moving source on the boundary of the weld line. This heat produced by the friction contact between the pin tool and the plates is concentrated locally, and propagates rapidly into remote regions of the plates by conduction according to Eq. (1) as well as convection and radiation through the boundary. It is assumed that the heat flux, \( q(r) \), is linearly distributed in the radial direction of the pin tool shoulder, and has the following form [4]:

\[
q(r) = \frac{12Qr}{\pi(d_o^3 - d_i^3)} \quad \text{for} \quad \frac{d_i^2}{2} \leq r \leq \frac{d_o^2}{2}
\]  

(2)

where \( d_o \) is the outside diameter of the pin tool shoulder, \( d_i \) is the pin diameter, and \( Q \) is the total heat input energy. In Eq. (2), the heat generated at the pin of tool is neglected because this heat is very small, e.g. in the order 2% of the total heat as reported by Russell and Sheercliff [8]. As such, in the analysis \( d_i = 0 \) in (2) was used.

On the boundary or the surfaces of the workpiece, convection and radiation in heat transfer are responsible for heat loss to the ambient. To consider such heat convection and radiation on all plate surfaces except for the bottom surface, the heat flux loss is evaluated by

\[
q_s = \beta(T - T_0) + \varepsilon B(T^4 - T_0^4)
\]  

(3)

where \( T_0 \) is the room temperature, \( \beta \) is the convection coefficient, \( \varepsilon \) is the emissivity of the plate surfaces and \( B = 5.67 \times 10^{-12} \text{Wcm}^{-2}\text{K}^{-4} \) is the Stefan–Boltzmann constant. In this calculation, \( \beta = 10 \text{Wm}^{-2}\text{K}^{-1} \) and \( \varepsilon = 0.17 \) are used for the 304L steel. The heat loss from the bottom surface is practically heat conduction from the workpiece to the support base plate. However, the complicated unknown contact resistance between the workpiece and support base presents difficulty in modeling. To circumvent the problem and simplify the analysis, we modeled this heat loss approximately using the heat flux loss by convection as

\[
q_b = \beta_b(T - T_0)
\]  

(4)

where \( \beta_b \) is a fictitious convection coefficient and an undetermined parameter. In addition, a latent heat is taken as 274 J/g to consider the possible phase transformation of the 304L steel during the FSW process.

Considering the boundary value problem of the heat transfer in the workpiece, if the total heat input energy, \( Q \), and the fictitious convection coefficient, \( \beta_b \), are known as in the heat transfer analysis for conventional fusion welding, one can numerically solve the differential equation (1) using the boundary conditions of (2)–(4). However, in the current FSW process the values of both \( Q \) and \( \beta_b \) are unknowns. Accordingly, an alternative analysis approach has to be investigated for the heat transfer analysis in the FSW. For this purpose, an inverse analysis method is developed in this work to numerically solve the boundary value problem for heat transfer in the workpiece. The analysis strategy is incorporated into the finite element analysis code—WELDSIM. The procedure of the inverse analysis method is outlined as follows:

1. Specify a set of temperature-dependent "guess" values for the fictitious convection coefficient, \( \beta_b \) in (4), which can be initially given as 10 times of the conventional convection coefficient for 304L steel.
2. Specify a "guess" value for the total heat input energy, \( Q \) in (2), produced by the contact friction between the tool shoulder and the plates. For example, it can be initially set \( Q = 1000 \text{W} \).
3. Solve the three-dimensional nonlinear differential equation (1) under the boundary conditions of (2)–(4) numerically using the finite element code—WELDSIM.
4. Output the temperature values during welding at the specific locations that the temperature histories were measured during the FSW process.
5. Compare the variations of temperature with time obtained from the finite element analysis and from the
2.3. Thermo-mechanical analysis

In the thermo-mechanical analysis, the incremental theory of plasticity is employed. The plastic deformation of the materials is assumed to obey the von Mises yield criterion and the associated flow rule. The relationship of the rate materials is assumed to obey the von Mises yield criterion.

\begin{align}
\dot{\varepsilon}_{ij} &= \frac{1}{E} \sigma_{ij} - \nu \delta_{ij} \dot{\varepsilon} + \lambda \delta_{ij} \left( \alpha \left( \frac{T - T_0}{T} \right) \right) \\
&= \frac{1}{E} \sigma_{ij} - \nu \delta_{ij} \dot{\varepsilon} + \lambda \delta_{ij} \left( \alpha \left( \frac{T - T_0}{T} \right) \right)
\end{align}

(5)

where \( E \) is the Young's modulus, \( \nu \) is the Poisson’s ratio, \( \alpha \) is the thermal expansion coefficient, \( \sigma_{ij} = \sigma_{ij} - \frac{1}{2} \sigma_{kk} \delta_{ij} \) are the components of deviatoric stresses, and \( \lambda \) is the plastic flow factor. \( \lambda = 0 \) for elastic deformation or \( \lambda < \sigma_0 \), and \( \lambda > 0 \) for plastic deformation or \( \sigma_{ij} \geq \sigma_0 \), where \( \sigma_0 \) is the yield stress and \( \sigma_{ij} = (3/2 \nu \sqrt{3})^{1/2} \) is the von Mises effective stress.

It is well known that the thermo-mechanical analysis for welding simulation using finite element method is extremely time-consuming. To reduce computational time and still maintain reasonable accuracy, many thermo-mechanical numerical analyses use a “cut-off temperature”, i.e., the mechanical properties above the cut-off temperature are assumed to maintain constant values [15]. Tekriwal and Mazumder [14] showed that the residual stresses from FEA have only small changes for carbon steels when the cut-off temperature varied from 600 to 1400 °C, but the computational time is significantly reduced if the cut-off temperature is assumed to be 900 °C (i.e., about two-thirds of 1400 °C—the melting temperature of 304L stainless steel) in the current numerical calculations to reduce unnecessary computational time.

All numerical computations are performed on PCs by using the FEA code—WELDSIM [17]. To further save computational time without loss accuracy of solutions, different time steps are used for the two un-coupled analyses, i.e. 0.1 s for the heat transfer analysis and 1.0 s for the stress-deformation analysis. Overall, the actual PC’s time in a typical simulation is about 3.5 min for the temperature analysis, and 120 min for the stress and deformation analysis.

2.4. WELDSIM—a welding simulation code

WELDSIM used in this work is a finite element analysis software specifically written for welding process simulation and developed at the University of South Carolina. The overall structure of this software can be found in Chao et al. [17] and Chao and Qi [18]. WELDSIM is a three-dimensional, nonlinear finite element computer code for the determination of transient, as well as residual, temperature, stress, strain and distortion of welded structures in a welding process including arc welding and FSW. Advanced features in the FEA code make WELDSIM robust and computationally efficient. Its efficiency has been widely verified through comparisons of computed results with experimental data [15,17,18].

In general, a heat transfer analysis and a subsequent thermo-mechanical analysis in WELDSIM are run sequentially or in a sequentially coupled manner. A moving heat source simulating the arc torch or FSW pin tool is modeled first to generate the temperature fields in the structure at various time steps during the welding process, and then the temperature history is used for the calculation of thermal stress and displacement fields in the structure during and after the welding process. The finite element formulation and solver in WELDSIM follow the standard procedures in computational mechanics. Details can be found in Chao and Qi [18]. Unique features in welding implemented into the code include: moving heat source model, complex heat input models for various types of heat sources, latent heat for simulating melting, dummy elements for simulating multi-pass welding, cut-off temperature to increase the computation speed, effective weight factor for elastic to plastic transition, flexibility in convergence criterion for both the heat transfer and stress analyses, both small and large deformation options for predicting distortion, and effect of welding fixture to the residual stress and distortion.

3. Numerical simulation and results

3.1. Finite element models

In the numerical simulation of the FSW for 304L stainless steel, it is assumed that the two plates are welded symmetrically during welding and after welding. The weld line is the symmetric line, and thus only half of the welded plate is modeled. The FEA mesh is shown in Fig. 3, which has...
two layers of elements in the thickness direction and 1800 eight-node brick solid elements and 3030 nodes in both the heat transfer analysis and the thermo-mechanical analyses. The smallest element used in the FEA model is in the region of the weld and has the dimension of 3.05 mm × 3.5 mm × 1.59 mm. It should be mentioned that in a mesh sensitivity study of welding simulation [17], an FEA mesh finer than used in this current work may generate nearly identical results, but the computational time increased significantly.

The weld zone in the FEA model is 304.8 mm in length, 10 mm in width on the top surface and 6.7 mm in width on the bottom surface, and is modeled by 500 eight-node brick solid elements. To simulate the change of material properties in the weld zone, the weld material properties are used in the weld zone behind the tool as the tool advances.

Since the fixture of clamped steel strips is released after the weld cools down to the room temperature, the residual stress in the weld will be redistributed after the fixture release. Consequently, the influence of the fixture release has to be modeled in the thermo-mechanical analysis. In the present FEA model, zero displacements are assigned to the finite element nodes in the clamped area of workpiece by the steel strips, shown as the shaded area in Fig. 1, during the welding and the cool down phases of the process. These fixed displacement boundary conditions are gradually released to become free in the last calculation of the elastic-plastic iterations at the room temperature to simulate the fixture release.

The material properties of 304L stainless used in the FEA computation are taken from the Metals Handbook of Brown et al. [19] and related articles of Kim et al. [20] and Reynolds et al. [21]. The thermal and mechanical properties are depicted in Fig. 4(a) and (b), respectively. The uniaxial, tensile stress-strain curves of 304L stainless steel and FSW welds for two rotational speeds of 300 and 500 rpm at the room temperature are given in Fig. 5 of Reynolds et al. [21]. The yield stresses for the base metal, 300 rpm weld and 500 rpm weld are 290, 370 and 440 MPa, respectively. The yield stresses of the two welds at high temperatures are assumed to vary with the same manner as the yield stress of the base metal and maintain a constant proportional factor with the yield stress of the base metal at the room temperature. For instance, the yield stress curve shown in Fig. 4(b) for the base metal is shifted up by 28%, i.e. \((370 - 290)/290 = 0.28\), for the yield stress of the 300 rpm weld material (see [3] for details). Use of different yield stresses in the base metal and in the weld can approximate the effect of weld property changes on the residual stresses. A constant Poisson’s ratio 0.3 is used since its change with temperature has little effect on the results in the computer simulation of welding. A constant plastic modulus of 2.8 GPa, which is determined from the experimental stress-strain curve [21], is also employed in all calculations to consider the effect of strain hardening on the residual stresses in the thermo-mechanical simulation. Note that the assumptions for the high temperature
material properties of the welds are necessary because these material properties are not readily available. Previous study by Zhu and Chao [15] has indicated that the yield stresses at all temperatures are the most important parameter in the thermo-mechanical simulation of welding.

3.2. Heat transfer simulation

Using the inverse analysis method as stated in Section 2.2, heat transfer simulations are performed for the FSW of 304L stainless steel. Only the last or the final numerical results are reported here. The variations of temperature from FEA and from measurement against time history at the middle of workpiece are shown in Fig. 5 for the rotational speed of 300 rpm. Fig. 5(a) compares the variations of temperature on the top surface at the thermocouple locations of 18, 21, and 26.5 mm, respectively, while Fig. 5(b) compares the variation of temperature on the butt end surface at the thermocouple locations of 12.7, 18, 21 and 27.5 mm, respectively.

Fig. 6 shows the distribution of temperature from FEA calculations along the transverse direction of the workpiece at welding time 83 s for the rotational speed of 300 rpm. The test results are also included in Fig. 6. The comparisons show that our FEA numerical results of temperature match with the experimental data very well. Moreover, since the plate is thin sheet, both the numerical and experimental results indicate that the difference of temperature between the top and bottom surfaces is very small. Fig. 7 illustrates the simulated temperature contour at welding time 111 s for the case having the tool rotational speed of 300 rpm. It is seen that the highest temperature during welding is within the tool shoulder and has the value between 900 and 1157 °C, which is much less than the melting temperature 1450 °C of 304L stainless steel. The 304L steel does not melt during the FSW, as from measurement and prediction. This confirms that the FSW of steels is also a solid-state jointing process, as generally accepted for FSW of aluminum alloys.
Similarly, Fig. 8 shows that the variations of temperature from FEA and from measurement against time history at the middle of workpiece for the rotational speed of 500 rpm. Fig. 8(a) compares the variation of temperature on the top surface at the thermocouple locations of 14, 17, 21, and 27 mm, respectively, whereas Fig. 8(b) compares the variation of temperature on the bottom surface at the thermocouple locations of 12, 15.5, 18, 21, and 27.5 mm, respectively. Once again, the comparisons show that our FEA results are in good agreement with the experimental data.

Comparing the data in Figs. 6 and 8, it can be seen that the maximum temperature at locations close to the weld line increases when the tool rotational speed increases from 300 to 500 rpm, and the difference of the maximum temperature at the same location is less than 100°C.

3.3. Energy transfer

In the case of the 300 rpm rotational speed, the heat input energy determined from the procedure of Section 2.2 is about 760 J/s, which is about 43.2% of the mechanical energy 1760 W calculated from the torque output of the machine. For the rotational speed of 500 rpm, the heat input energy determined and used in the thermal simulation is 970 J/s, which is about 44.7% of the total mechanical energy 2240 W output from the machine. From these data, it appears
that close to 50% of the total mechanical energy from the FSW machine is transformed to increasing the temperature of the workpiece during FSW. The other 50% energy may be dissipated in (a) the large plastic deformation of the weld material, the clamped fixture strips, and the support plates, and (b) the heat transferred to the tool and the machine head. Note that this percentage of mechanical energy transferred into heating the workpiece is less than the value 75–80% as reported by Chao et al. [11] for aluminum alloy 2195.

3.4. Thermo-mechanical simulation

With the numerical results of temperature fields, we can then carry out the thermo-mechanical analysis using the FEA code, WELDSIM. In the calculation of residual stresses, we have included two special steps: (a) modeling the FSW fixture release when the temperature cooled down to the room temperature, and (b) modeling the overmatched material property of the welds using actually measured yield stress instead of using the same base material property. The details of numerical modeling are described in Section 3.1.

As in arc welding, the primary residual stress in FSW occurs in the weld and is in the longitudinal or weld-line direction. The variations of the longitudinal residual stress along the transverse direction at the middle section of workpiece are shown in Figs. 9 and 10 for the rotational speed of 300 and 500 rpm, respectively. The FEA results of the longitudinal stress before fixture release and after fixture release are plotted in these two figures. They are compared with the test data which are measured at \( z = 0.55, 1.3, 2.05, \) and 2.8 mm.
4. Conclusions

To study the variations of transient temperature and residual stresses in friction stir welding of 304L stainless steel plates, detailed three-dimensional nonlinear thermal and thermo-mechanical simulations are performed for the FSW process using the finite element analysis code—WELDSIM. Two welding cases with tool rotational speeds of 300 and 500 rpm are considered. From the analyses, we can summarize the results as follows:

(1) Based on the experimental data of transient temperature history at several specific locations during the FSW for 304L stainless steel, an inverse analysis method for thermal numerical simulation is developed. Results show that due to unknown heat energy input from the process, this inverse analysis method is unique and effective for the calculation of temperature field in the FSW.

(2) For the two welding cases, the numerically determined temperature fields match well with the experimental data. The maximum temperature during the FSW is at the weld line and within the tool shoulder. The difference of the maximum temperature at the same location between the rotational speed of 300 and 500 rpm is less than 100°C. The maximum temperature determined from the simulation is between 900 and 1000°C, which is significantly less than the melting temperature of 304L stainless steel at 1450°C.

(3) Using the numerically calculated temperature field, the residual stress in the friction stir welded plate is then determined in a subsequent three-dimensional elastic-plastic mechanical simulation. Fixture release is also modeled. The numerically calculated residual stresses are consistent with the experimental data. The difference of residual stress between the two cases, tool rotational speeds of 300 and 500 rpm, is small.

(4) The residual stress in the welds after fixture release decreases significantly as compared to those before fixture release. Thus the fixture release in the FSW must be considered in the computer simulation for the determination of residual stresses.

(5) Close to 50% of the total mechanical energy measured from the FSW machine is transformed to increasing the temperature of the workpiece during FSW. Relative to the 75–80% obtained from FSW of aluminum alloy 2195 [11], this value is certainly much less. Whether these values are from the difference in both the tool and the workpiece materials or the potential errors in the numerical modeling warrant further investigation. More detailed studies are underway at the University of South Carolina and results will be reported later.

Acknowledgements

This work is supported by US Air Force Research Laboratory (AFRL) Contract No. F33615-96-D-5835, DO#0083, and US Office of Naval Research (ONR) Agreement No. N00014-00-0-3-0008 which MTS Systems Corporation is the Prime Contractor. The encouragement from Dr. Kumar Jata of AFRL and Dr. George Yoder of ONR is greatly appreciated. Dr. Wei Tang at the University of South Carolina performed the welding and the temperature measurement. The residual stress test data were provided by Drs. Hank Peak and Thomas Gnaupel-Herold of the National Institute for Standards and Technology, Gaithersburg, MD, USA.

References


