Numerical procedure for residual stresses prediction in friction stir processing of Al-4.5%-10% TiC metal matrix composite

Pili kumari Sahoo, Prashant KumarMishra
Department of Mechanical Engineering, College of Engineering Bhubaneswar, Biju Patnaik University of Technology, Bhubaneswar, India

Mail ID: baishya_nayan84@gmail.com

Abstract

The prediction of residual stresses is a relevant and, under many points of view, still open issue for a proper welding process design. In the present paper a 3D FE model, with general validity for different joint configurations, was used to simulate the Friction Stir Processing(FSP) of butt joints through a single block approach. The model is able to predict the residual stresses by considering thermal actions only, thanks to a new time efficient approach. A good agreement between calculated and experimentally measured data was found; the effectiveness of the presented numerical procedure was evaluated by comparing the calculation times of the proposed method with the ones of already known FE approaches

Introduction

One of the most promising and investigated welding process of the last years is Friction Stir Processing (FSP), a solid state process invented and patented in 1991 by TWI. The process mechanics is governed by the action of a specially designed rotating tool which is inserted into the adjoining edges of the sheets to be welded with a proper tilt angle, and then moved all along the joint. The tool produces frictional and plastic deformation heating in the welding zone although the joining of the blanks occurs in solid state conditions; in other words no melting of material is observed during the process. As the tool moves along the welding line and rotates, the material is forced to flow around it in a quite complex flow pattern: such effect was extensively investigated, especially as the process was initially proposed; Rhodes in [1], Liu in [2] and Guerra in [3] tried to highlight the material flow following different approaches. In fact, one of the first steps of the research on FSP was the investigation of the process mechanics and the influence of the most relevant process parameters on the mechanical properties of the welded joints. A few geometrical and technological parameters strongly influence the process: the tool geometry affects both the metal flow and the heat generation due to the friction forces work, while the tool rotating speed and feed rate combination determines the heat flux conferred to the joint during the process. Liu investigated the effects of process parameters on the dimension of different micro-structural zones in the transverse section of the welded joints [2], Shigematsu et al. [4] considered the effect of the material characteristics and Lee et al. [5] showed the mechanical effect of the grain refinement in the nugget zone of the joints.

More recently a few papers focused on the surface integrity and, consequently, on the analysis of the residual stress field in FS welded joints. John and, Bussu and Irving [6] initially correlated the fatigue life of the joints to the residual stress state showing the positive effects induced by the latter but, in turn, underlined that residual stresses cause distortions in the welded plates. Another relevant contribute was given by Peel et al. [7], who carried out synchrotron X-ray measurements to obtain the residual stress field in AA5083 friction stir welded sheets at the varying of the tool feed rate. Similar investigations were developed by Staron et al. [8] and Preve´y and Mahoney [9]. Finally, Reynolds et al. [10] extended the investigation to stainless steel joints.

In the last decade a consistent work was performed on the numerical simulation of FSW in order to develop a proper numerical model able to aid the design process. Two main approaches were considered, namely the use of analytical models, reproducing the thermal flux generated during the process, and of thermo-mechanical simulations through FE models. One of the first attempts were made by Song and Kovacevic [11] and Chao et al. [12] who proposed numerical models taking into account the heat generated by both friction-forces and material deformation work.

The model developed by Chen and Kovacevic in [13] uses the commercial FEM software ANSYSTM: a thermo-mechanically coupled Lagrangian finite element model, incorporating temperature and multilinear strain hardening, is used for the three-dimensional model- ing of the solid structures. The temperature evolution is obtained modeling the tool action thermal effects as a moving heat source. Although no strain rate dependent material model was considered, the authors were able to investigate the effect of the heat moving source on the workpiece material.

More recently Fourment in [14] uses an Arbitrary Lagrangian Eulerian formulation implemented into the Forge3s software with a splitting approach and an adaptive remeshing scheme based on error estimation. The first step is the computation of the thermo- mechanical fields in a Lagrangian approach. Then the mesh velocity is calculated minimizing the estimated error. Finally, the step variables and the updated mesh are transferred with a remapping step. 50,000 elements with minimum size of 0.5 mm are used for the plate. In this way the contact interactions between the tool and the workpiece were investigated in details and the importance of such issue in the numerical modeling of the FSW processes is underlined.

The models developed by Mandal et al. [15] focused on the difficulties of considering the high strain rates and temperatures involved in the process resulting in a complex numerical problem involving the non-linear material behavior. The simulation of the entire FSP process was performed using the Johnson–Cook elasto- plastic constitutive law available in the finite element code ABAQUS^{TM}. Remeshing was extensively used within an ALE approach with the aim to eliminate excessive element distortion which could lead to a premature termination of the simulation. In a different way, some of the authors [16] presented a continuous 3D coupled thermo-mechanical FE model in which the tool–workpiece interaction is considered. The model will be briefly presented in the next section of the paper.

As far as the numerical prediction of residual stresses is regarded, Chao et al. [12], Chen and Kovacevic [13] and Khandkar et al. [17] developed thermal analyses for stainless steel and aluminum alloys. These analyses were based on properly tuned analytical models with the aim to highlight the residual stress state due to the thermal history the material locally experiences during the process. A few considerations can be developed on the latter papers: first of all the local mechanical action of the tool, and in particular of the tool pin, is not considered. Only the analytically calculated thermal flux is taken into account obtaining a sort of macro-effect of the process on the material. Moreover, the adopted thermal models, which describe the heat flux due to the tool action, are always axial-symmetric; basically, no effect of the asymmetric material flow in FSW processes is considered introducing a strong assumption for the calculation of the residual stress state.

Starting from the developed FEM model [16], the authors set up a procedure able to overcome a few severe limitations of the previous models [18]. The actual thermal histories, resulting from the coupled thermo-mechanical analysis developed taking into account the effect of the asymmetrical material flow, were calculated for each node of the model and passed to a further elasto-plastic FEM model thus obtaining the residual stress state. It should be noticed that by considering the histories of temperature calculated as described above, a more realistic representation of the residual stresses fields can be presented even if the contribution of the mechanical action of the tool pin on the residual stresses themselves is not considered. Finally, the effect of the residual stress on the fatigue life was estimated by fatigue crack propaga- tion (FCP) tests. In particular, the fatigue behavior of a crack approaching the friction stir weld was compared with that of a numerical prediction in which the numerical residual stresses of the weld are considered. Since compressive residual stresses reduce the growth rate, it is expected that the fatigue life of FWS joint results higher than the one of base material [19].

In the present paper a new approach is used to evaluate the

residual stresses distributions in FSP processes considering a single computational environment, allowing residual stresses prediction regardless to the joint geometrical configuration and using just one FE software. The main aim of the utilized approach is to overcome the main drawback deriving from the use of an elasto-plastic FE model for the simulation of the whole FSW process and the prediction of residual stress in the joint, that is, especially for large dimension joint of industrial interest, the extremely long computational time. Residual stresses calculated through the proposed approach were compared with those measured during the experimental tests. Calcu- lation times for different FE based approaches were collected inorder to assess the effectiveness of the presented procedure.

2.The proposed problem

2.1 FSW experiments

In the present study the FSP of AA6060-T4 aluminum alloy was performed butt joining two 300 ~ 100 mm^2 sheets, 4 mm in thickness. During experiments and numerical simulations, the following tool design was utilized: pin height equal to 3.6 mm, conical pin diameter of 5 mm with cone angle of 301 and shoulder diameter equal to 15 mm (see Fig. 1). As far as the process technological parameters are regarded, tool shoulder penetration into the sheets, tool rotating speed and tilt angle were kept constant and equal to 0.2 mm, 500 rpm and 21, respectively. Three levels of advancing velocity, i.e. 100, 225 and 325 mm/min were selected. Residual stress measurements were carried out by means of the cut compliance technique described by Prime in [20] on the base of Schindler's weight functions in [21] in order to verify the correctness of the numerical procedure. In particular three repeti- tions were developed for each case study; in the following paragraphs the average obtained values will be reported.

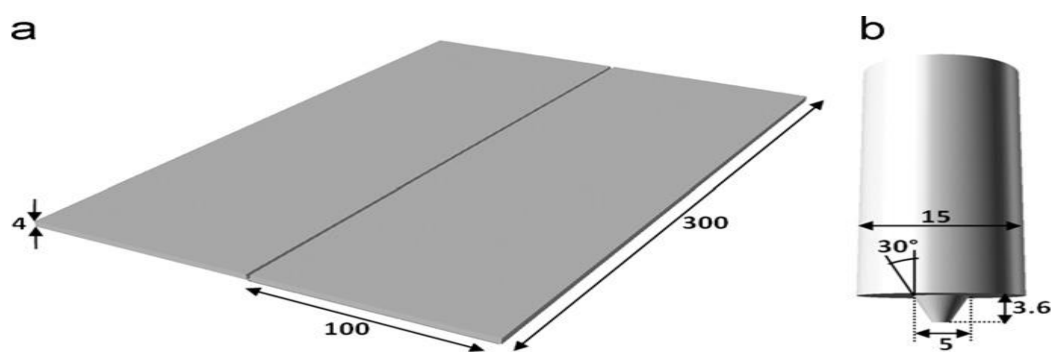

Fig. 1. Sketch of the used (a) joint and (b) tool geometries

2.2

FEM model

Despite the significant recent advances in the numerical simu- lation of the FSW process, most of the models often have several limitations in either the representation of the geometries, or the material behavior, or the boundary conditions. In the present research a numerical procedure was set up to determine the residual stress field starting from the FEM model presented in [16]. This fully 3D FEM model is thermo-mechanically coupled and assumes a rigid-viscoplastic material behavior. A unique feature of this model is the representation of the sheet seam (abutting edges) as a continuum. This continuum hypothesis avoids the numerical instabilities that may result from the discontinuities present at the edge of the two sheets.

The commercial FEA software DEFORM-3D™ [22], a Lagrangian implicit code designed for metal forming processes, was used to model the FSP process. The use of an implicit code versus an explicit one is a sort of inescapable choice being the latter better suited in order to correctly predict temperature evolutions and stress states. The following is a summary of model equations governing the investigated mechanical and thermal equilibrium.

As far as the mechanical analysis is regarded, a rigid-viscoplastic material model with Von Mises yield criterion and associated flow rule was used. As it is well known, the rigid-viscoplastic finite element formulation is based on the variational approach. The actual velocity field is determined from the stationary value of the variation of governing functional p, as described in detail in [16]:The workpiece was modeled as a rigid viscoplastic material, and the welding tool is assumed rigid. This assumption is reasonable as the yield strength of the sheet (i.e. aluminum alloy) is significantly lower than the yield strength of the tool (tool steel or carbide). However, it should be observed that such an assumption for the workpiece material behavior denies the possibility to get residual stresses at the end of the process simulation; indeed, an elasto- plastic material model must be used to get the final residual stress state in the worked material. As far the numerical modeling of FSW processes is regarded, this choice results in extremely long computational times – as shown in detail in the next paragraphs – due to the complex non-linearity of the mechanical problem and the severe gradients of the main process field variables as strain and strain rate around the tool action area.During the FSP process, the workpiece is clamped at its ends and supported at its bottom by a back plate; these boundary conditions were modeled in the FE analysis. The FSP simulation is then divided into two stages: (i) sinking stage and (ii) Processing (advancing) stage. In other words, FSP is modeled from its initial state to the end, i.e. when the tool is removed from the clamped blanks. During the sinking stage, the tool moves down vertically with a rotating speed; then, during the welding stage, it moves along the welding line joining the two workpieces. The tool is tilted of a small angle with respect to a vertical axis and at the end of the sinking stage a proper dwelling time is considered to assure a sufficient heating of the blanks material for the subsequent welding process.In order to develop a proper tuning of the numerical model an inverse approach was used starting from experimental tempera- ture history and vertical forces data [16].For the thermal characteristics of the considered AA6060-T4 aluminum alloy, the following values were utilized: thermal conductivity k1¼120 [N/(s 1C)] and thermal capacity c ¼2.4 [N/ (mm2 1C)]. No variation of k1 and c with temperature was taken into account; this assumption linearizes the thermal problem and results in a better convergence. A constant interface heat exchange coefficient of 11 [N/(mm s 1C)], based on previous experience, was utilized for the tool–sheet contact surface. As the value of the heat transfer coefficient is regarded, a typical convection coefficient in calm air equal to 0.02 [N/(mm s 1C)] was selected. A preliminary sensitivity analysis for different interface heat exchange coefficient values confirmed that there was no significant variation in the workpiece calculated temperature.As briefly described before, a rigid-viscoplastic temperature and strain rate dependent material model was used as follows:s ¼ 2:03 ~ 103T— 0:488ðe_Þ0:0027ðeÞ0:124 ½MPa]

ð3Þwhere the temperature is expressed in K and the strain rate in s—1. It should be noticed that the material behavior was input in the utilized software in form of tabular data starting from both literature data [23] and in house experiments. Eq. (3) is then obtained through linear regression just to represent the material

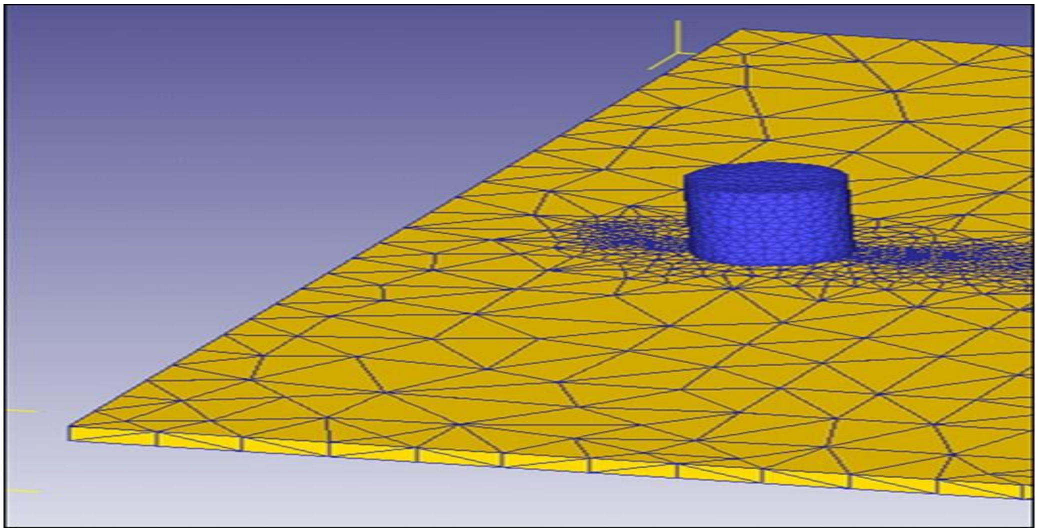

Fig. 2. Sketch of the model at the end of the sinking phase.

behavior at the varying of temperature, strain and strain rate. The used data were also in good agreement with those used by Khandkar et al. [17].

For the modeling of the workpiece, a ‘‘single block’’ continuum model (sheet blank without a gap) was used to avoid contact instabilities due to the intermittent contact at the sheet–sheet and sheet–tool interfaces. In particular, a single sheet was considered and the rotating tool, once completed the sinking phase, was moved forward welding the crack left behind the pin as it advances along the welding line.

The sheet blank was meshed with about 24,000 tetrahedral elements with single edges between 0.5 and 2.5 mm. A non-uniform mesh, characterized by density windows all along the welding line and adaptive remeshing were adopted in order to place smaller elements close to the tool. A remeshing referring volume was identified all along the tool feed movement (Fig. 2). The choice of tetrahedral elements instead of other elements, e.g. brick elements, is driven by the need to have reasonable computational times. Proper accuracy is obtained through the increase of the number of elements, especially around the tool pin. In this way a faster and at the same time accurate convergence is assured.

Experience in previous FEM simulations shows that a coarser mesh leads to incorrect results and frequent software stops; on the other hand, a finer mesh results in unaffordable computational time without significant improvement in the results accuracy. The contact between the workpiece and the back plate was modeled by locking the contact surface nodes.

2.3 Developed FE approach

It is widely recognized that numerical simulation represents a valuable aid in the FSW processes design. In particular, the determi-nation of residual stress, leading to the welded joint fatigue life prediction, has a strategic importance for most of the applications in the aeronautical, aerospace, automotive and naval industries. Unfortunately, its extremely high computational cost still represents an important drawback: as a matter of fact, depending on the joint dimensions and utilized process parameters, a thermo-mechanical coupled analysis with elasto-plastic material behavior including the mechanical action of the tool may take weeks, thus losing most of its potential benefits, especially as far as industrial applications are regarded.

The developed approach, able to effectively cut down the computational cost of simulations, is based on the idea of splitting the process simulation into two different simulations. The process is first simulated from the very beginning, i.e. the tool sinking, to its end, i.e. the tool removing from the blanks, with a rigid-viscoplastic material model as described in the previous paragraph. In this way a relatively fast convergence is reached although no residual stress can be obtained. At the end of the process simulation a new database, characterized by the mesh utilized for the first step of the rigid-viscoplastic simulation and including, for each node, the temperature history that will be experienced during the welding in the form of boundary conditions, is created by a specific routine developed and embedded in the simulation software. The latter operation includes several sub-steps described more in detail in the following (Fig. 3): the desired time interval, e.g. 0.33 s in this study, is selected and a set of files containing all nodal temperatures for the selected steps, i.e. the selected process times, is automatically created. It is worth noting that, during FSW process simulation, the heavy deformations induced by the disrupting mechanical action of the tool result in frequent remeshing steps and, in this way, the nodal temperatures are referred to several different meshes. Then, for each selected process time, nodal temperature information referred to a particular mesh is added to the first step mesh through a logarithmic interpolation and converted in thermal time dependent boundary conditions for each of the nodes of the first mesh. This operation is automatically developed regardless to the joint dimension and/or morphology, i.e. butt joints, lap joints, T joints, etc. Once the new database is generated, a simulation characterized by elasto-plastic material behavior is run. The workpiece thus experiences the thermal evolution resulted from the previously developed thermo-mechanically coupled analysis with no further tool mechanical action. The last statement can be accepted based on the basic assumption – well established in the scientificcommunity – that the residual stresses mainly depend on the thermal histories experienced by the workpiece [24].During this latter simulation the tool object was disabled and the workpiece conserved all the boundary conditions reproducing the actual clamping fixtures. This new simulation, denying the tool mechanical action and avoiding the step convergence difficulties typical of elasto-plastic simulations with large deformations, is characterized by very short computational times.

The final step of the numerical procedure consisted in the simulation of the joint cooling down back to the environment temperature. This last simulation was in turn divided into two further different sub-simulations since during the initial period of the actual cooling stage, the workpiece is still clamped to the welding machine while, in the second phase of cooling simulation, when the joint is removed from the welding machine, the boundary constraints are released and just three nodes of the workpiece mesh are kept fixed in order to avoid any rigid movement of the workpiece itself. At the end of this final step the residual stress state in the workpiece is obtained.

A preliminary attempt to obtain residual stresses by separating the thermo-mechanical rigid-viscoplastic simulation from the elasto-plastic one was presented by the some of the authors in [18]. In particular, the data coming from the DEFORM™ simulation were manually transferred to a further elasto-plastic FE model using a brick type mesh in Abaqus/Standard environment for a coupled temperature-displacement analysis. Such interpola- tion procedure, carried out in Matlab™ environment, had to be repeated for every single FSW process simulation extracted step. Since during the welding process simulation the mesh topology changes several times due to the necessary remeshing steps, the whole procedure was time consuming. Additionally, as the two FE environments, namely Deform and Abaqus, are not directly com- patible to each other, for every single joint dimension or config- uration, i.e. butt joints, lap joints, T joints, etc., a dedicated routine had to be written resulting in a further increase in the time required for the calculation of the residual stress.

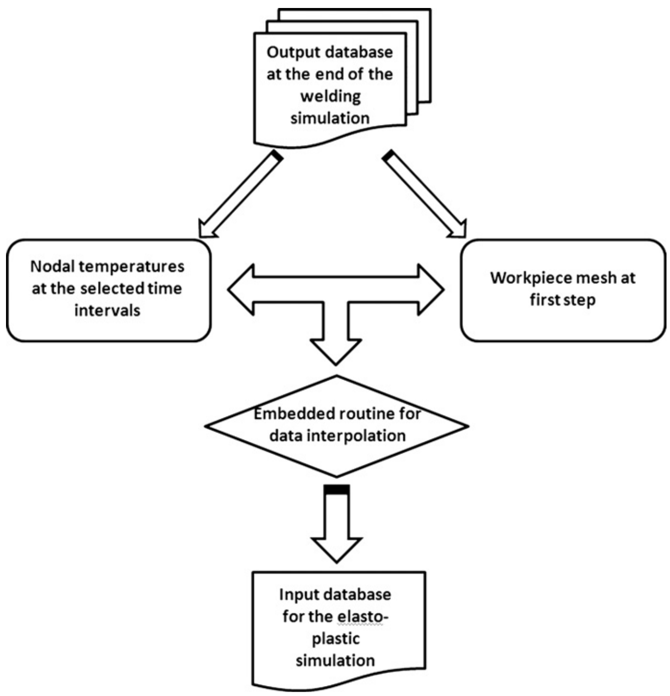

Fig. 3. Schematic representation of the proposed approach.

3. Results

As described in the previous paragraph, the first step of the developed procedure was the simulation of the welding process with a rigid-viscoplastic material model, based on the assumption that, due to the extremely large deformations occurring, elastic deformation can be neglected with no significant loss in the results accuracy. In this way the nodal temperatures and their evolution during the FSW process were calculated. Fig. 4 shows the tem- perature evolution on the top surface of the workpiece for process conditions characterized by tool rotating speed of 500 rpm and tool feed rate of 100 mm/min. During the sinking stage a temperature peak of about 340 1C is reached while the typical temperature comet profile, with a maximum level of about 428 1C, is found once the process reached its steady state. The obtained temperatures were previously verified, during the tuning stage of the model, through experimental data [16].

Then, once collected the temperature values for every node and selected time step, the data are properly interpolated in the first step mesh. In this way, for each node, a temperature history in the form of thermal boundary condition is obtained. Fig. 5 presents, for the 500 rpm and 100 mm/min case study, a typical temperature evolution curve obtained for a node laying in the advancing side at a distance of 6 mm from the welding line and 2 mm from the bottom of the joint that is at mid thickness.

Finally, once all the simulation phases of the elasto-plastic model are completed, that are thermal load, initial cooling down keeping mechanical constraints and final cooling down to room

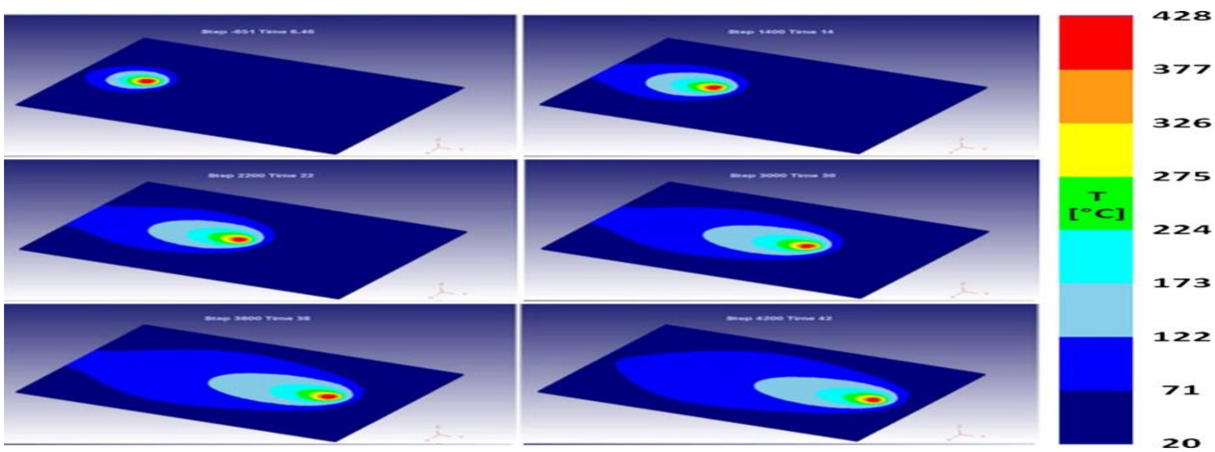

Fig. 4. Workpiece temperature evolution during the welding process simulation—500 rpm and 100 mm/min case study.

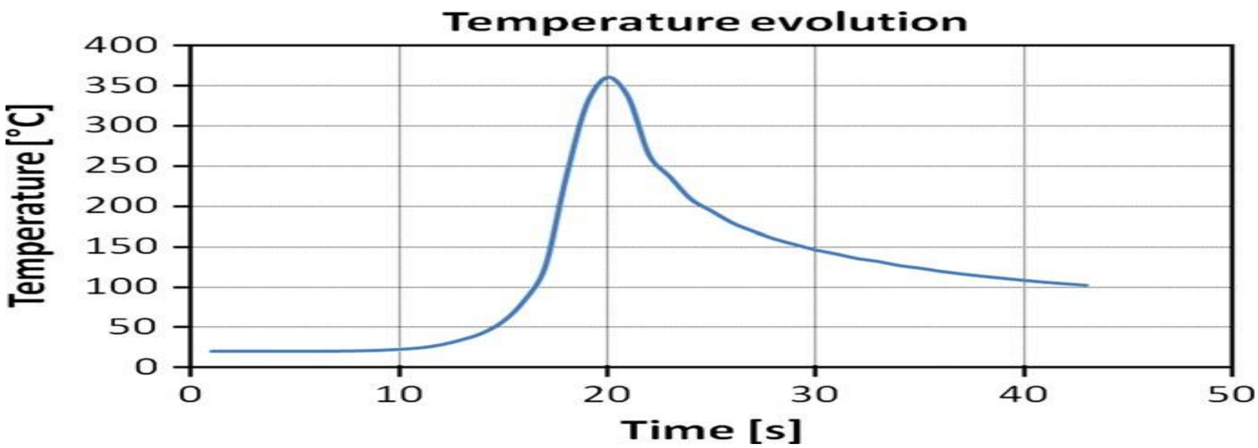

Fig. 5. Temperature evolution of a generic node laying in the advancing side 6 mm from the welding line at mid-thickness—500 rpm and 100 mm/min case study.

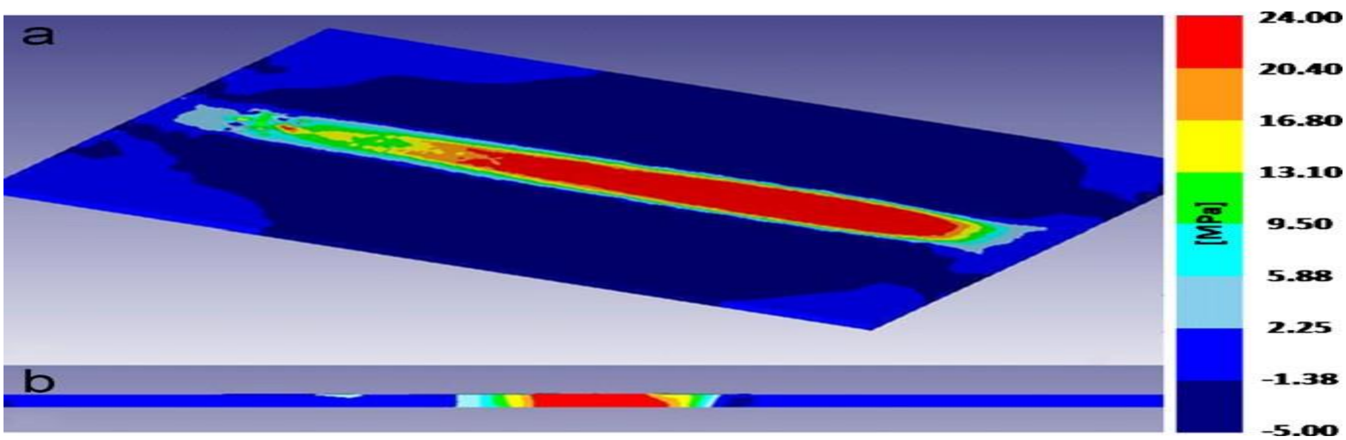

Fig. 6. Longitudinal residual stress distribution (a) on the top surface and (b) in a transverse section of the joint—500 rpm and 225 mm/min case study.

temperature with constraints released, the residual stress profiles can be generated. In Fig. 6 the longitudinal residual stress profiles, both on the joint top surface and in a transverse section taken at mid-length, i.e. 150 mm from the joint edge, are shown. It should be observed that the longitudinal stress distribution is the most interesting to investigate for FSW processes, as the largest residual stresses are expected parallel to the processing line, i.e. parallel to the tool trajectory [25]. As it can be seen from the figure, when the process reaches its steady state, a tensile stress is found at the center of the joint all along the welding line and including the area interested by the tool action, while a compressive state can be observed at the periphery of the joint.

In order to assess the correctness of the obtained results, longitudinal residual stress calculated values have been compared with the ones experimentally measured by the cut compliance method. Figs. 7 and 8 show this evaluation in a transverse section taken at mid thickness for the 500 rpm and 225 mm/min and the 500 rpm and 325 mm/min case studies, respectively. A few obser- vations can be made from the above figures. The two calculated curves are in good agreement with the correspondent experimen- tally measured values, in terms of both curve shape and maximum values reached. In particular, as it was expected, larger longitudinal residual stress values are found for the 500 rpm and 225 mm/min case study with respect to the 500 rpm and 325 mm/min one. The

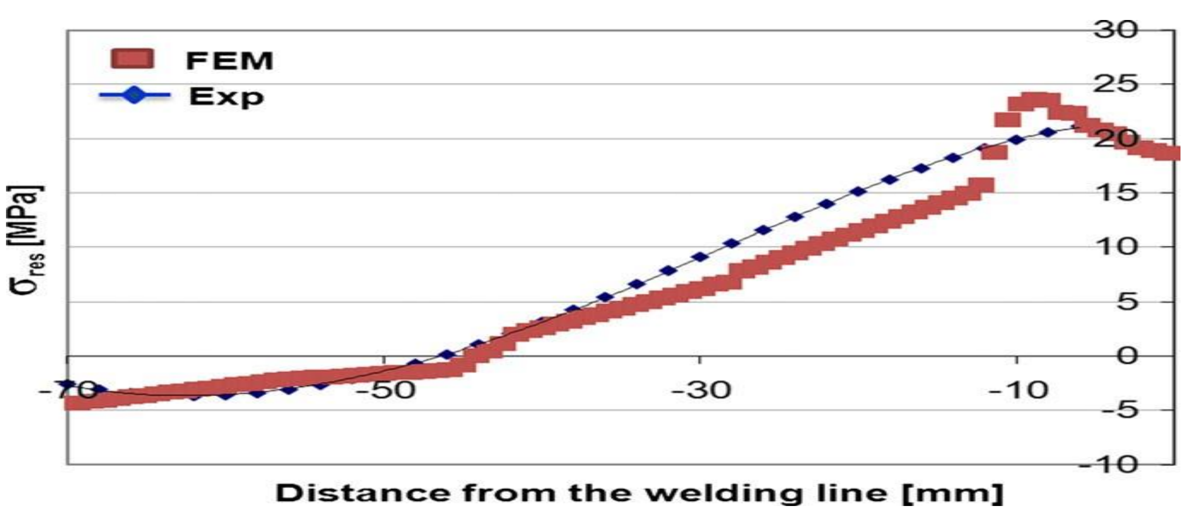

Fig. 7. Experimental vs. numerical residual stress distribution in a transverse section at mid-thickness: 500 rpm and 225 mm/min case study.

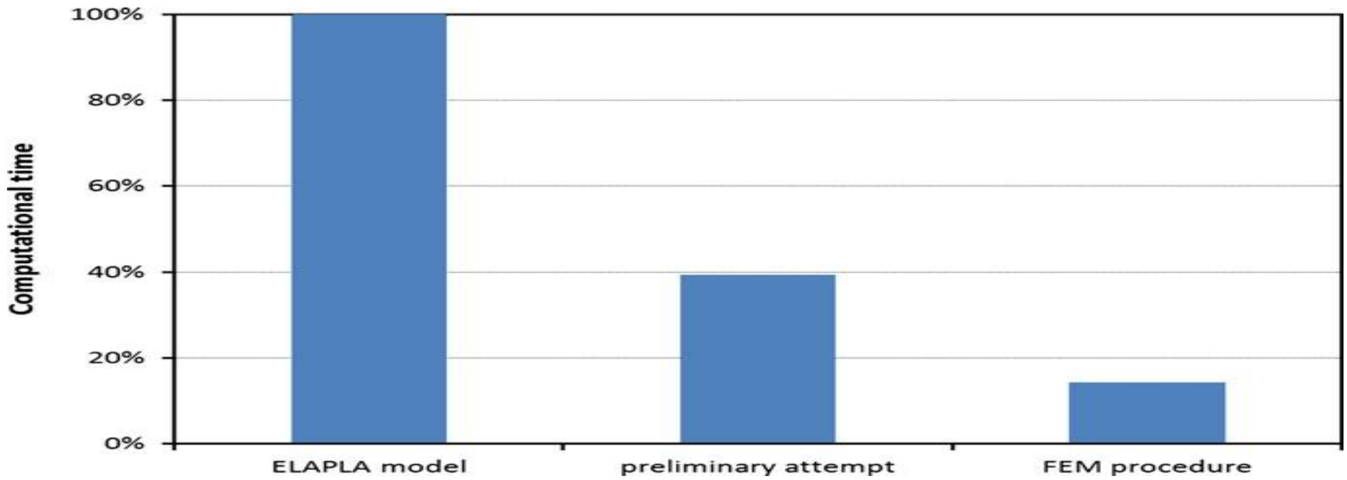

Fig. 9. Computational times for the three considered approaches.

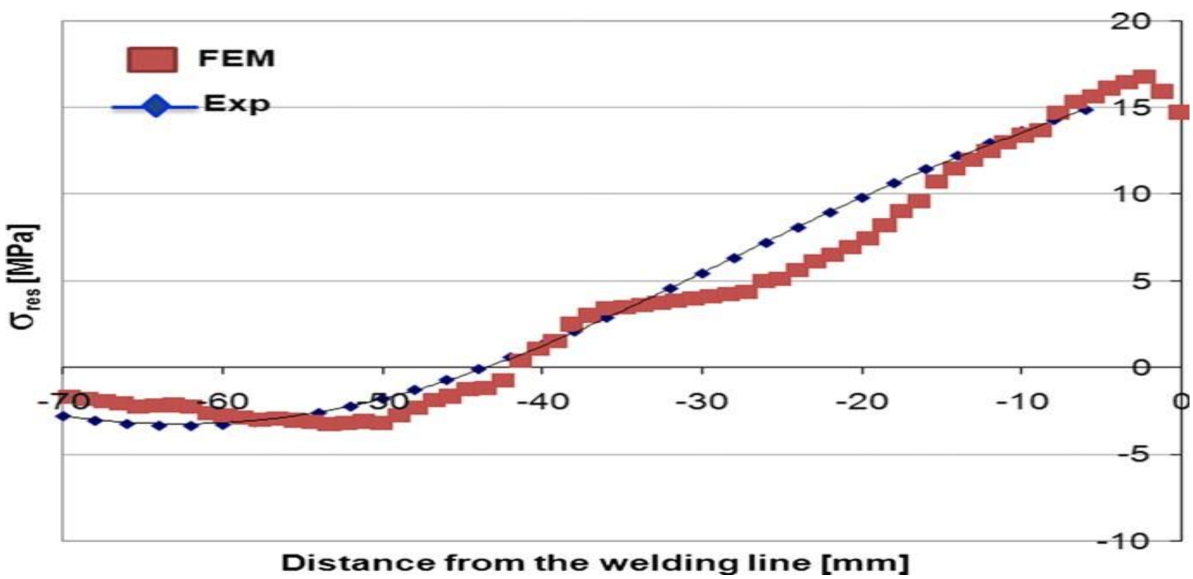

Fig. 8. Experimental vs. numerical residual stress distribution in a transverse section at mid-thickness: 500 rpm and 325 mm/min case study.

former is characterized by a larger thermal contribution conferred to the joint and, as a consequence, larger temperatures during the welding process. In turn, the maximum residual stress value, equal to about 30 MPa, was observed both numerically and experimen- tally for the 500 rpm and 100 mm/min case study. However, the numerical model tends to underestimate the actual residual stress value at a distance from the welding line in the range between 10 and 30 mm, which corresponds to the Heat Affected Area (HAZ) of the transverse section. It should be noticed that the simulation performed under similar conditions, using an elasto-plastic mate- rial model and including the mechanical action of the tool, gave residual stress results within 2% with respect to the ones calculated through the proposed procedure and represented in Figs. 7 and 8. Finally, in order to quantitatively evaluate the effectiveness of the proposed procedure, the total computational time required – from the setting up of the first simulation to the obtainment of the residual stress

– has been measured for the three considered approaches (Fig. 9). In particular, the time required to perform a single thermo-mechanical simulation with elasto-plastic material including the tool action was considered as benchmark and marked as 100%. Accordingly, the time required for the other two approaches – the preliminary attempt of splitting the problem described in Section 2.3 and the procedure

presented in this paper – was measured as about 40% and 15% of the reference time. The choice of showing times as percentage of the fully elasto-plastic model was driven by the need to get rid of the difference that may exist in calculation times depending on the available processing power. The time required for the fully elasto-plastic model, utilizing an up to date multiprocessor computer workstation, was 14 days in this study. A sit can be seen, although an additional simulation is needed, a dramatic reduction in the computational time is obtained with the new proposed procedure, allowing about 85% time savings by splitting the problem into two separate simulations. Regarding the preliminary attempt, the time necessary to manually transfer the data to the new simulation software and to write, in a further programming environment, a proper subroutine, which must be dedicated to each different joint dimension and morphology, was included and significantly contributes to increase the total time. It is a worthy notice that the difference in computational time between the considered approaches can further increase as the dimension or the complexity of the model increase.

4.Conclusions

In the paper a new numerical procedure for the prediction of the residual stress distributions in Friction Stir processes is presented. The procedure, based on both a rigid-viscoplastic thermo- mechanically coupled model and an elasto-plastic thermo-mechani- cally coupled one, has as main target the dramatic reduction of computational time, in fact this represents, at the moment, the most severe drawback of the already known fully elasto-plastic models characterized by comparable results accuracy. The FSP of butt joint of two AA6060-T4 aluminum alloy sheets of relevant dimensions was chosen to test the procedure.

The obtained calculated distributions, as compared to experi- mentally measured ones, show a good capability of the model to predict longitudinal residual stresses.

Additionally, the obtained results are very similar to the ones obtained with the single fully elasto-plastic model, which, in turn, required an extremely longer computational time.

The proposed procedure is then able to allow significant time savings with respect to the other tested approaches, resulting in a useful research and industrial tool for extensive numerical investigations.

References

[1] C.G. Rhodes, M.W. Mahoney, W.H. Bingel, R.A. Spurling, C.C. Bampton, Effects of friction stir welding on microstructure of 7075 allum, Scr. Mater. 31 (1) (1997) 69–75.

[2] G. Liu, L.E. Murr, C.S. Niou, J.C. McClure, F.R. Vega, Microstructural aspects of the friction-stir welding of 6061-T6 aluminum, Scr. Mater. 37 (3) (1997) 355–361.

[3] M. Guerra, C. Schmidt, J.C. McClure, L.E. Murr, A.C. Nunes, Flow patterns during friction stir welding, Mater. Charact. 49 (2) (2002) 95–101.

[4] I. Shigematsu, Y.J. Kwon, K. Suzuki, T. Imai, N. Saito, Joining of 5083 and 6056—T4 allum. alloys by friction stir welding, J. Mater. Sci. Lett. 22 (5) (2003) 353–356.

[5] W.B. Lee, Y.M. Yeon, S.B. Jung, The tImprov. of Mech. prop. of friction-stir-welded A356 Al alloy, Mater. Sci. Eng. A A355 (2003) 154–159.

[6] G. Bussu, P.E. Irving, The role of residual stress and heat affected zone prop. on fatigue crack propagation in friction stir

welded 2024-T351 allum Joints, Int. J. Fatigue 25 (1) (2003) 77–88.

[7] M. Peel, A. Steuwer, M. Preuss, P.J. Withers, Microstructure, Mech. prop. and residual stresses as a function of welding speed in allum. AA5083 friction stir welds, Acta Mater. 51 (16) (2003) 4791–4801.

[8] P. Staron, M. Kocak, S. Williams, Residual stresses in friction stir welded Al sheets, Appl. Phys. A: Mater. Sci. Process. 74 (Suppl. II) (2002) S1161–S1162.

[9] P. Preve´ y, M. Mahoney, Improved fatigue perform of friction stir welds with low plasticity burnishing: residual stress design and fatigue perform assess- ment, Mater. Sci. Forum 426–432 (4) (2003) 2933–2940.

[10] A.P. Reynolds, Wei Teng, T. Gnaupel-Herold, H. Prask, Structure, prop., and residual stress of 304L stainless steel friction stir welds, Scr. Mater. (2003) 48.

[11] M. Song, R. Kovacevic, Thermal modeling of friction stir welding in a moving coordinate system and its validation, Int. J. Mach. Tools Manuf. 43-6 (2003) 605–615.

[12] Y.J. Chao, X. Qi, W. Teng, Heat transfer in friction stir welding, Exp. Numer. Stud., Trans. ASME 105 (2008) 138–145.

[13] C. Chen, R. Kovacevic, Thermomechanical modelling and force analysis of friction stir welding by the finite element method, Proc. Inst Mech. Eng., Part C: J. Mech. Eng. Sci. 218 (5) (2004) 509–520.

[14] M. Assidi, L. Fourment, S. Guerdoux, T. Nelson, Friction model for friction stir welding process simulation: calibrations from welding experiments, Int. J. Mach. Tools Manuf. 50 (2) (2010) 143–155.

[15] S. Mandal, J. Rice, A.A. Elmustafa, Experimental and numerical investigation of the plunge stage in friction stir welding, J. Mater. Process. Technol. 203 (2007) 411–419.

[16] G. Buffa, J. Hua, R. Shivpuri, L. Fratini, A continuum based FEM model for friction stir welding—model development, Mater. Sci. Eng. A 419/1–2 (2006) 389–396.

[17] M.Z.H. Khandkar, J.A. Khan, A.P. Reynolds, M.A. Sutton, Predicting residual

thermal stresses in friction stir welded metals, J. Mater. Process. Technol. 174 (1–3) (2006) 195–203.

[18] L. Fratini, G. Macaluso, S. Pasta, Residual stresses and FCP prediction in FSW through a continuous FE model, J. Mater. Process. Technol. 209 (2009) 5465–5474.

[19] L. Fratini, S. Pasta, A.P. Reynolds, Fatigue crack growth in 2024-T351 friction stir welded joints: longitudinal residual stress and micro-structural effects, Int. J. Fatigue 31 (3) (2009) 495–500.

[20] M.B. Prime, Residual stress measurements by successive extension of a slot: the crack compliance method, Appl. Mech. Rev. 52 (2) (1999) 75–96.

[21] H.J. Schindler, Experimental determination of crack closure by the cut compliance technique, ASTM Special Technical Publication, 1999, pp. 175–187.

[22] DEFORM, Deform-3DTM, User manual, 2003.

[23] Y.V.R.K. Prasad, S. Sasidhara, Hot working guide: a compendium of process, in:

Y.V.R.K. (Ed.), Maps, 1997.

[24] G. Buffa, L. Fratini, S. Pasta, R. Shivpuri, On the thermo-mechanical loads and the resultant residual stresses in friction stir process operations, Ann. CIRP 57/1 (2008) 287–290.

[25] T. Li, Q.Y. Shi, H.K. Li, W. Wang, Z.P. Cai, Residual stresses of friction stir welded 2024-T4 joints, Mater. Sci. Forum 580–582 (2008) 263–266.