Available online at www.sciencedirect.com

ScienceDirect

Procedia CIRP 133 (2025) 460–465

www.elsevier.com/locate/procedia

20th CIRP Conference on Modeling of Machining Operations

Development and optimization of a finite element model with remeshing and Lagrangian formulation for the simulation of high deformation manufacturing processes

Ignacio-Manuel Valdivia-Maldonado[a,*], Ainara Oruna[a,b], Gorka Ortiz-de-Zarate[a], François Ducobu[c], Guénaël Germain[b], Pedro J. Arrazola[a]

[a]Mondragon Unibertsitatea, Faculty of Engineering, Loramendi 4, 20500, Arrasate-Mondragón, Spain
[b] LAMPA, Arts et Métiers Institute of Technology, 2 boulevard du Ronceray BP 93525, 49035 Angers, France
[c] UMONS Research Institute for Materials Science and Engineering, University of Mons, Place du Parc 20, Mons, 7000, Belgium

* Corresponding author. Tel.: +34 943 79 71 60; fax: +34 943 79 15 36. *E-mail address:* imvaldivia@mondragon.edu

Abstract

High deformation manufacturing processes, such as forming and machining, are complex physical phenomena involving severe thermo-mechanical and chemical loads. Traditional industrial-scale empirical methods involve high tooling and preparation costs and long lead times before manufacturing, which is undesirable in modern industry. The use of predictive models helps to reduce these weaknesses. Finite Element Method (FEM) models are a useful, reliable and cost-effective tool for studying manufacturing processes. Several approaches have been used to model these processes with the FEM. The Lagrangian formulation with implicit time integration scheme is the most widely used because of its reliability. However, element distortion due to severe plastic deformation and chip separation, in the case of machining, has always been a major concern of this approach. This paper therefore presents the development of a customizable and optimized FEM model with Lagrangian formulation and remeshing technique that solve the mesh distortion problem. The model was developed using the general-purpose software Abaqus/Standard commanded by Python scripting. The remeshing criterion is based on the relative plastic deformation at each load increment controlled by two subroutines working together UVARM+URDFIL. A forming problem was selected to optimize the mesh size and number of remeshings with the goal of reducing the simulation time. Then, the proposed model was compared to Lagrangian models without remeshing and Arbitrary Lagrangian-Eulerian (ALE) formulation. The model was also experimentally validated demonstrating improvements over other approaches and formulations, and laying the foundation for further development, such as applying it to the machining process.

Keywords: Finite Element Method (FEM), Abaqus, Remeshing, Manufacturing, Python scripting

1. Introduction

Predictive models can be integrated into process planning systems to improve productivity and enhance product quality. They can also be applied in adaptive control for manufacturing processes, reducing or eliminating the need for trial-and-error approaches [1]. The Finite Element Method (FEM) is the most commonly used approach for engineering simulations, particularly in solid mechanics and high-deformation problems [1, 2, 3, 4]. The FEM has different formulations tailored to address specific concerns regarding material behaviour and responses. Four dominant continuum-based approaches have been used in research for modelling manufacturing processes, which are the Lagrangian (LAG) model [5, 6], Eulerian (EU) model [7], Arbitrary Lagrangian-Eulerian (ALE) model [8] and Coupled Eulerian-Lagrangian (CEL) [9]. For LAG and EU models, Lagrangian or Eulerian meshes are the only mesh types applied for simulation, regardless of the meshing control techniques, while ALE and CEL models use both Lagrangian and Eulerian meshes.

Lagrangian mesh, in which meshes share the same coordinates as the material points and deform together, is the most widely used type of mesh in FEM simulations of manufacturing processes. However, for LAG formulation

10.1016/j.procir.2025.02.079

modelling severe plastic deformation problems, the excessive distortion of meshes is one of the most dominant issues that lead to a failure of calculation. To overcome this problem, there are two main techniques when using the LAG approach, the element deletion technique [10], in which important information is lost, and the remeshing technique [11, 12, 13], which proposes a solution for mesh distortion without the need of special geometrical or physical criteria. This method can automatically change the mesh sizes and shapes within a deformation area to achieve the complete simulation of a process, and excessive distortion of elements could be fully avoided. In the remeshing technique, when the part elements satisfy predefined critical conditions, a new mesh is generated and all the state variables of the nodes of the old mesh are interpolated to the nodes of the new mesh.

However, the mentioned remeshing technique has certain disadvantages such as: i) the complicated algorithms and coding that must be programmed, ii) diffusion of the results when interpolating the state variables from the deformed mesh to the new mesh and iii) determining the criteria for triggering the remeshing. This last point is particularly important, as it directly influences the accuracy of the results and the calculation time [3]. The general-purpose software Abaqus is the preferred by researchers to develop their codes and models due its capabilities and robustness [14]. As for the remeshing technique, few studies have developed their own remeshing code using Abaqus [12], and even fewer have optimized the remeshing technique [15]. However, no optimization of the technique has been performed considering computation time, result accuracy, mesh size, and the remeshing criterion.

Specialized FEM software for machining, such as Advant-Edge [16, 17, 18] and DEFORM-2D/3D [19, 20, 21], which incorporate remeshing techniques, have been widely used by researchers in recent years, making significant contributions. However, extensive investigations [2, 3, 14] have highlighted notable limitations that constrain future developments and research. Their limitations lie in that only few can simulate the entire process chain, as is the case of the commercial FEM-software Simufact.forming [22] (e.g. forging, thermal treatment, machining, shot peening, etc.), they have very limited control by the user, impossibility to simulate any material, since they have limited material libraries, and limited types of materials and elements, although they have the possibility of accessing modules through coding, their implementation is complex. In addition, there is no control over the remeshing technique, criteria for trigger remeshing, remeshing criteria, types of elements in the new mesh, quality of the elements, specific areas to be remeshed, etc.

To leverage the high performance of specialized machining modelling software and address its limitations, this paper presents the development and optimization of a FEM model using an updated Lagrangian formulation with a remeshing technique for simulating manufacturing processes. The model was developed using the general-purpose software Abaqus/Standard commanded by Python scripting. The remeshing criterion is based on the relative plastic strain controlled by two subroutines working together UVARM+URDFIL. To optimize the model, a forming problem was chosen to refine the mesh size and number of remeshings, aiming to reduce simulation time. The proposed model was then compared with standard Lagrangian models without remeshing and with the Arbitrary Lagrangian-Eulerian (ALE) formulation, followed by an experimental validation.

2. Development of the model

2.1. Python scripts and user-subroutines

The remeshing tool was developed using the general-purpose software Abaqus/Standard 2022 and comprises a main script that commands the simulation and seven additional scripts, each tasked with specific functions. These scripts are executed sequentially from the main script to automate the entire simulation process, beginning with the initial MESH-0 model and culminating in the generation of a remeshing animation. A detailed flow chart of the simulation with remeshing tool is shown in Fig. 1. The main script is executed from the initial Abaqus/CAE MESH-0 model and it begins by creating the input file (.inp) for the MESH-0 model and executing it alongside the UVARM and URDFIL user-subroutines. When the critical threshold value for the relative plastic strain (Δ PEEQ) is reached, the simulation stops, generating the output database file (.odb) for MESH-0. The script then enters a loop, sequentially invoking subsequent scripts that extract data from the previously generated output database file (*.odb) and proceed to generate the next MESH-*i* (where *i* represents the remeshing iteration) output database file. The user must specify the threshold value within the subroutines file. Finally, after completing the remeshing loop, the script generates an animation of a user defined variable, utilizing data from all the generated output databases (.odb).

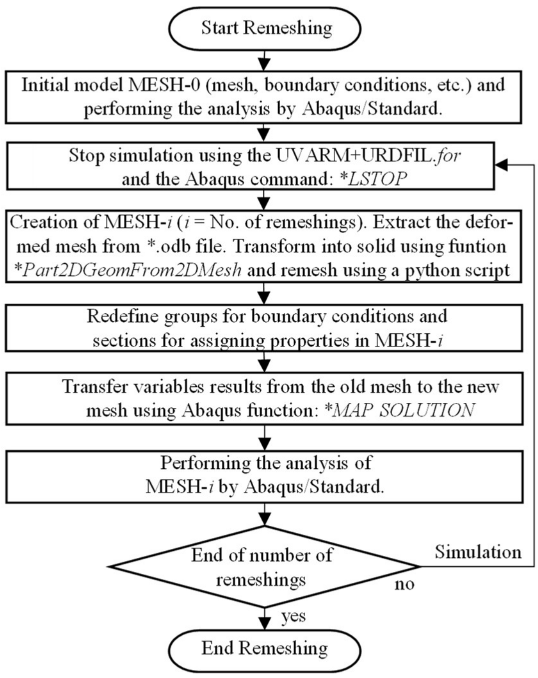

Fig. 1. Flow chart of the remeshing routine.

2.2. Modelling of a high deformation manufacturing process

To carry out an optimization of the developed model and to perform numerical and experimental validations, a high deformation manufacturing process was selected. Uniaxial compression test of Ti6Al4V cylindrical specimens was simulated. The geometry of the specimen and the boundary conditions of the FEM model is shown in Fig. 2, and consists of a cylindrical billet 9 mm long, with a radius of 3 mm (in accordance with ASTM E9-09 and ASTM E209-00 (2010) Standards), compressed between flat and rigid dies. The simulation finishes when the length was reduced by 60%. To reduce calculation times, the chosen finite element model is axisymmetric and includes only the upper half of the billet (see Fig. 2), since the central surface of the billet is a plane of symmetry. In the FE model elements of type CAX4T, 4-node bilinear displacement and temperature that allow for fully coupled temperature displacement analysis, were used. The simulations were carried out at 293K with a strain rate of 1 s^{-1}.

2.3. Simulation plan

The simulation plan was divided into two parts. In the first part, an optimization analysis was performed based on the threshold value of the relative plastic strain, which is directly related to the number of remeshings and mesh size. In the second part, a comparison of the optimized model with other models with different time integration schemes and formulations was carried out. All the simulations were run on a computer with an Intel(R) Xeon(R) Gold 6142 with two processors running at 2.60 GHz using 128 GB of RAM.

2.3.1. Optimization of the remeshing model

An optimization study was conducted to determine the optimal threshold value (Δ PEEQ) and mesh size for the model. The input parameters of the model are presented in Table 1. The thermo-viscoplastic behaviour of the Ti6Al4V ELI billet was modelled with Johnson-Cook law as it was found to correctly represent the behaviour of a similar material [23, 24]. However, no ductile failure model was implemented. First, the threshold value (Δ PEEQ) was set at 0.2 and numerical tests of the upsetting of a cylindrical billet with different mesh sizes (mesh seeds) in the range of 0.08 mm to 0.7 mm were performed. Secondly, the mesh size was set to 0.1 mm and tests carried out with different threshold values (Δ PEEQ) in the range of 0.08 and 1.0. For each test, the calculation time and the relative error were determined. The relative error was calculated using the control simulation results as a reference, which was performed with the smallest threshold value (0.08) and the smallest mesh size (0.08 mm).

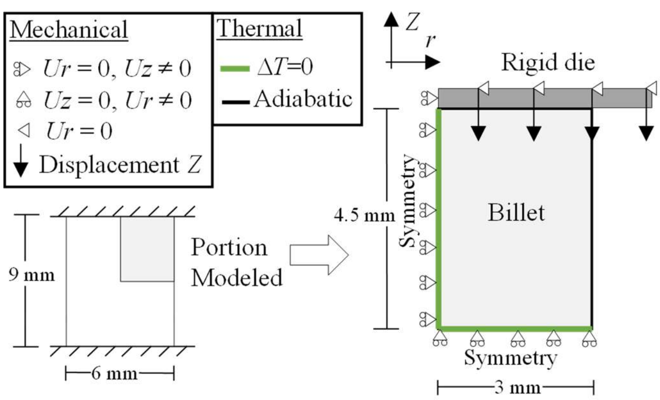

Fig. 2. Boundary conditions of the FEM model

Table 1. Input parameters for the FEM model [24].

Type of analysis	Coupled temperature-displacement
Total analysis time [s]	1
Discretization, mesh seed [mm]	0.08 - 0.7
Elements	CAX4T, CAX3T
Thermo-physical properties workpiece Ti6Al4V	
Density [kg/m^3]	4430
Young's Modulus [GPa]	117.2 (293K) 82.7 (923K)
Poisson ratio	0.33
Thermal conductivity [W/m/K]	6.9 (293K) 18 (1223K)
Expansion [1/K]	9.1 e^{-6} (293K) 1.1 e^{-5} (1088K)
Specific heat [J/kg/K]	520 (293K) 763 (923K)
Johnson-Cook parameters workpiece Ti6Al4V	
A [MPa]	352
B [MPa]	440
C	0.0083
n	0.42
m	1
T_m [K]	793
T_0 [K]	293

2.3.2. Comparison with other time integration schemes and formulations

The numerical comparison with other integration times and formulations was divided into two stages. In the first stage, fundamental variables (plastic strain, temperature, and von Mises stress) and mesh quality were compared between the optimized model and other integration times and formulations. The same element size was used for all simulations, the models listed below were run:

- ia Abaqus/Standard without remeshing.
- ib Abaqus/Explicit without remeshing.
- ic Abaqus/Explicit dynamics with Arbitrary Lagrangian Eulerian (ALE) with 20 node reordering per iterations (adaptive meshing).

In the second stage, the computing times were compared between the optimized model and the ALE formulation. Since both approaches involve changes in the mesh during FEM analysis, the following tests were conducted:

- iia Abaqus/Explicit dynamics with Arbitrary Lagrangian Eulerian (ALE) with 5 node reordering per iteration (adaptive meshing).
- iib Abaqus/Explicit dynamics with Arbitrary Lagrangian Eulerian (ALE) with 10 node reordering per iteration (adaptive meshing).
- iic Abaqus/Explicit dynamics with Arbitrary Lagrangian Eulerian (ALE) with 20 node reordering per iteration (adaptive meshing).

3. Methodology of the experimental tests

Uniaxial compression tests were carried out on the Gleeble 3500 machine employing Ti6Al4V ELI Ø 6 x 9 mm cylindrical samples. Two repetitions were carried out to ensure the robustness of the results. During the tests, the temperature was controlled by K-type thermocouples to correct the adiabatic heating using the method presented in [25]. Thin graphite foils were placed between the anvils and the specimen to reduce friction. Fig. 3 is a schematic representation of the compression set-up. Tests were carried out at 293K with a strain rate of 1 s^{-1}.

4. Results and discussion

4.1. Optimization of the remeshing model

As can be seen in Fig. 4a the error of plastic strain and temperature as a function of mesh size is not very critical, showing small error variations. However, the model shows a high sensitivity of the von Mises stress as a function of mesh size. Fig. 4b shows that the threshold value affects the plastic strain and in particular the von Mises stress significantly. It is therefore critical to carefully select the mesh size to ensure robust prediction of von Mises stress and to determine the optimal threshold value for accurately predicting both plastic strain and von Mises stress. Although the model was validated for room temperature and a strain rate of 1 s^{-1}, it can also be applied to represent machining conditions (high strain rates and temperatures) due to the type of element used (CAX4T displacement and temperature) which allows for such calculations.

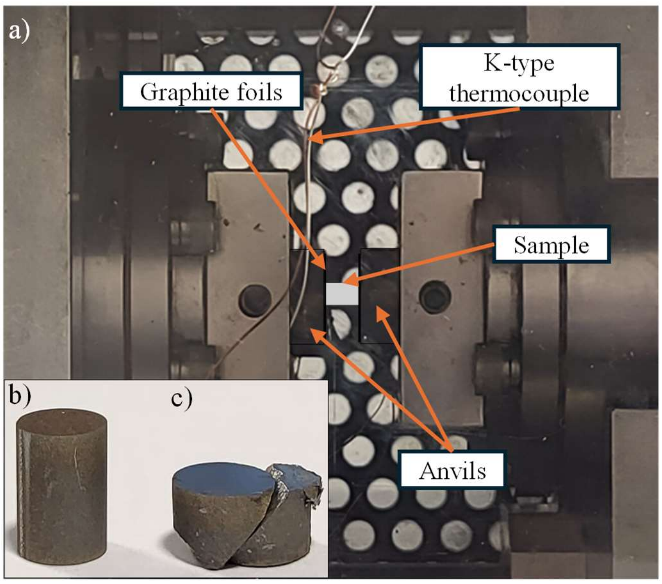

Fig. 3. a) Experimental set-up, b) uncompressed sample, c) compressed sample.

For the optimal simulation point, two criteria were considered: minimising prediction error and selecting the smallest mesh size, as a fine mesh will be used for machining simulation. This ensures remeshing times for forming are comparable to machining simulation times.

Therefore, based on the results obtained, it can be concluded that for the model analysed the optimum mesh size seeds is 0.15 mm and the optimum threshold value is 0.2 (equivalent to a number of remeshings of 20), since from the graphs (Fig. 4) this parameters showed an stabilization of the error value and the best results in terms of the relationship between low computation time and low variable error.

In summary, the optimization tests carried out showed that the remeshing modelling can be optimized and the outcomes are sensitive to the threshold value (directly related to the number of remeshings) and mesh size.

4.2. Comparison with other time integration schemes and formulations

4.2.1. Comparison of fundamental variables

Fig. 5 shows the mesh at the end of the simulation for plastic strain for each model run. As can be seen, the mesh in a, b and c does not respect the rigid surface and even exits above it, which implies a numerical error, due to the severe deformation.

In general, the results with remeshing Fig. 5d presents a better distribution of the variables obtained (temperature, plastic strain, von Mises, etc.) and a structured and good quality mesh throughout the forming process, with not numerical errors, despite its large deformation.

4.2.2. Comparison of computing time

Table 2 shows the results of the computing times for the optimized remeshing model and ALE formulation models. The table shows that the calculation times are very sensitive and depend on the requirements imposed on the model. In a general view, the computation times of the optimized model are fully comparable to the computation times obtained for the model with ALE formulation, even with the one with more numerical requirements (20 node reordering per iteration, Table 2 iic). It should be noted that even with these requirements do not solve the problem of the mesh respecting the rigid surface, Fig. 5c.

Based on the results it can be said that the optimized model results in lower meshing error, a more homogeneous distribution of the analysed variables and a reduced computation time.

Table 2. Comparison of computing times

FEM formulation		Computing time [min]
iia	ALE formulation (frec= 10, Sweeps=5), mesh seeds 0.15 mm.	41
iib	ALE formulation (frec= 10, Sweeps=10), mesh seeds 0.15 mm.	75
iic	ALE formulation (frec= 10, Sweeps=20), mesh seeds 0.15 mm.	83
opt.	Abaqus/Standard, LAG formulation, 20 Remeshings, mesh seed 0.15 mm.	78

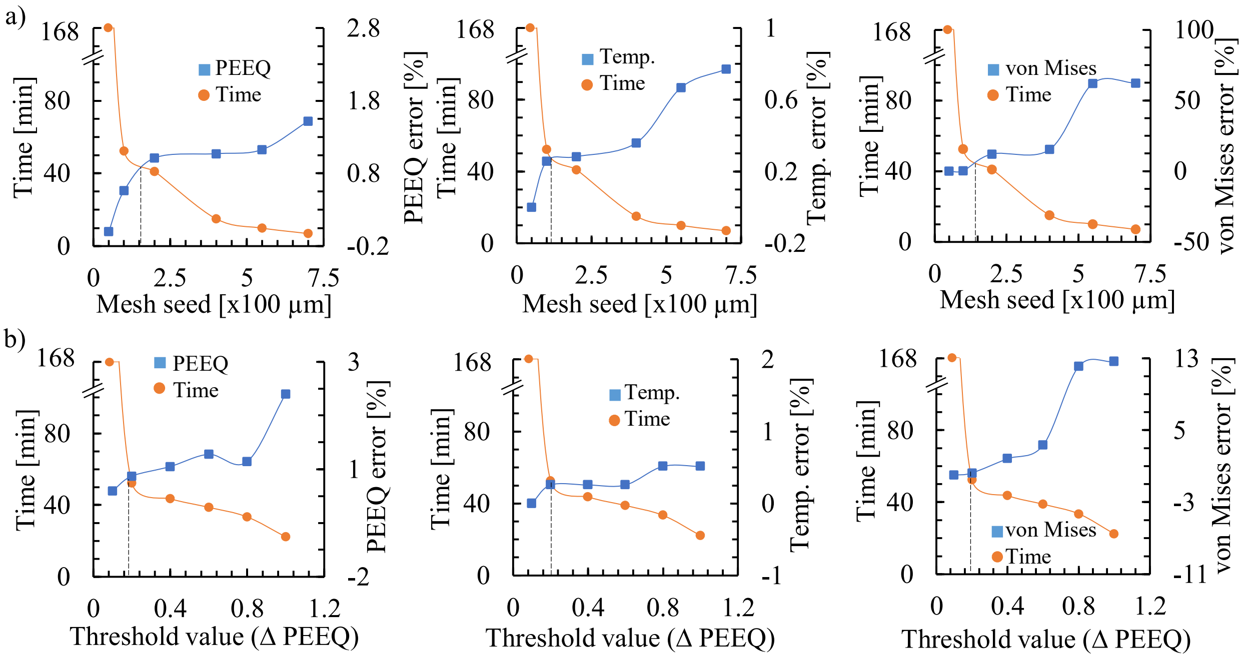

Fig. 4. Results of the optimisation study. Relationship between the relative error of plastic strain (PEEQ), temperature, and von Mises stress with respect to a) mesh size (mesh seeds), and b) threshold value (Δ PEEQ).

4.3. Experimental tests

As can be seen in Fig. 6a the load vs. displacement curve shows good agreement up to a compression of 1 mm, then both curves follow the same upward trend with an average error of 8%. This may be due to the strain hardening parameter of Johnson-Cook's law used is for normal Ti6Al4V [24] and the tests were performed with Ti6Al4V ELI. Regarding the true stress vs. plastic deformation curve, Fig. 6b, there is an agreement between the curves throughout their development with an average error less than 2%.

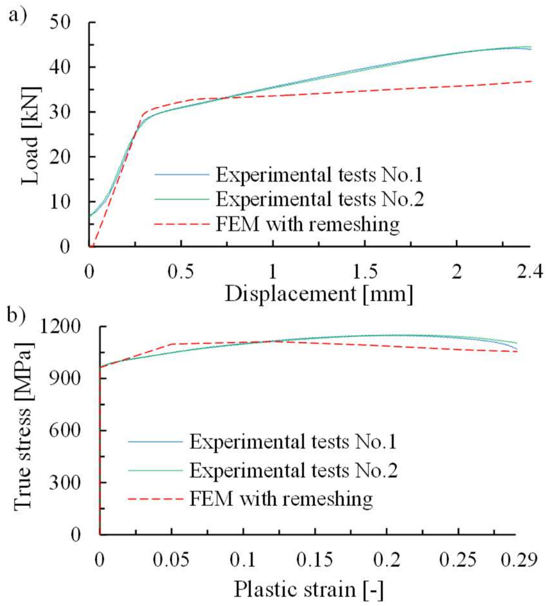

Fig. 6. a) Load vs Displacement, b) Stress vs Plastic Strain.

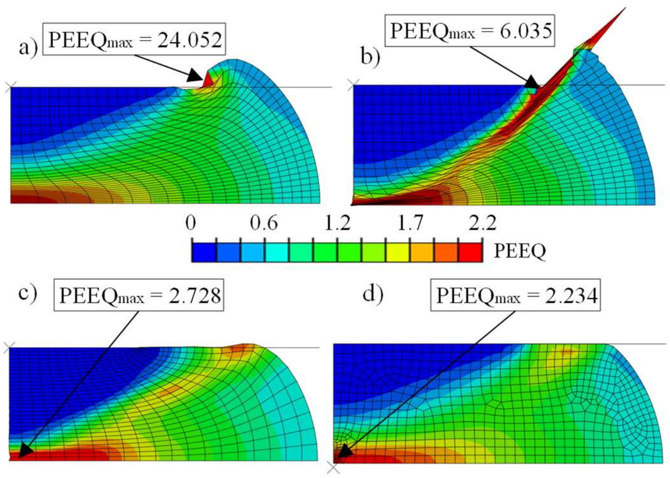

Fig. 5. FEM Results of plastic strain (PEEQ). a) Abaqus/Standard without remeshing (ia). b) Abaqus/Explicit without remeshing (ib). c) Abaqus/Explicit with Arbitrary Lagrangian Eulerian (ALE) formulation with 20 node reordering by iteration (ic). d) Optimized model develop in Abaqus/Standard with remeshing technique (20 remeshings and mesh seeds equal to 0.15 mm).

5. Conclusions and future outlook

In this paper, a novel finite element model with Lagrangian formulation in the general-purpose software Abaqus/Standard with user-subroutines and Python coding to command the remeshing routine is presented. The main highlights of the study are the following:

- Optimization tests of the remeshing model for the upsetting of a cylindrical billet were conducted to reduce computational time while maintaining result accuracy. The tests demonstrated that the model can be effectively optimized, revealing high sensitivity of the von Mises stress to mesh size and significant sensitivity of both plastic strain and von Mises stress to the threshold value (Δ PEEQ) which is directly related to the number of remeshings. Therefore selecting these both numerical input parameters (mesh size and threshold value) accurately is crucial to ensure precise computation of these variables.

- Comparison with other time integration schemes and formulations has demonstrated that the optimized remeshing model delivers superior mesh quality, more robust results, and reduced computational time.
- The FEM results show good agreement with the experimental tests, with an average error of 8% for load-displacement and less than 2% for true stress-plastic strain.

Having demonstrated the effectiveness of the developed remeshing model, its implementation in machining processes (2D orthogonal cutting) is proposed as a future direction. This will involve the scripting to adapt the mesh by zones of interest (make a finer mesh in the cutting zone), the integration of more advanced material models to capture phenomena such as complex flow stress behaviour and ductile failure. Additionally, specific friction laws tailored for machining will need to be integrated to enhance the robustness and accuracy of the predictions [26].

Acknowledgements

The authors hereby thank the support of the projects TAILORSURF (PID2022-139655OB-I00), nG24 (KK-2024/0001), HEGEMA (ZL-2023/00755) and Grupo de Investigación de Mecanizado de Alto Rendimiento IT1443-22.

References

[1] P. J. Arrazola, T. Ozel, D. Umbrello, M. Davies, I. S. Jawahir, “Recent advances in modelling of metal machining processes,” CIRP Annals - Manufacturing Technology, vol. 62, pp. 695–718, 2013.

[2] M. Sadeghifar, R. Sedaghati, W. Jomaa, V. Songmene, “A comprehensive review of finite element modeling of orthogonal machining process: chip formation and surface integrity predictions,” International Journal of Advanced Manufacturing Technology, vol. 96, pp. 3747–3791, 6 2018.

[3] J. M. Rodrıguez, J. M. Carbonell, P. Jons´en, “Numerical methods for the modelling of chip formation,” Archives of Computational Methods in Engineering, vol. 27, pp. 387–412, 4 2020.

[4] M. E. Korkmaz, M. K. Gupta, “A state of the art on simulation and modelling methods in machining: future prospects and challenges,” Archives of Computational Methods in Engineering, vol. 30, no. 1, pp. 161–189, 2023.

[5] F. Zanger, P. Bollig, V. Schulze, “Simulative investigations on different friction coefficient models,” Procedia CIRP, vol. 58, pp. 140–145, 2017.

[6] W. Cheng, J. C. Outeiro, “Modelling orthogonal cutting of Ti-6Al-4V titanium alloy using a constitutive model considering the state of stress,” The International Journal of Advanced Manufacturing Technology,vol. 119, no. 7-8, pp. 4329–4347, 2022.

[7] C. C. Wang, “Finite element simulation for forging process using euler’s fixed meshing method,” in Materials Science Forum, vol. 575, pp. 1139–1144, Trans Tech Publ, 2008.

[8] M. Movahhedy, M. Gadala, Y. Altintas, “Simulation of the orthogonal metal cutting process using an arbitrary lagrangian–eulerian finiteelement method,” Journal of materials processing technology, vol. 103, no. 2, pp. 267–275, 2000.

[9] F. Ducobu, P.-J. Arrazola, E. Rivière-Lorphèvre, G. O. De Zarate, A. Madariaga, E. Filippi, “The cel method as an alternative to the current modelling approaches for Ti6Al4V orthogonal cutting simulation,” Procedia CIRP, vol. 58, pp. 245–250, 2017.

[10] B. Wang, Z. Liu, “Shear localization sensitivity analysis for johnson–cook constitutive parameters on serrated chips in high speed machining of ti6al4v,” Simulation Modelling Practice and Theory, vol. 55, pp. 63–76, 2015.

[11] T. Marusich, M. Ortiz, “Modelling and simulation of high-speed machining,” International Journal for numerical methods in engineering, vol. 38, no. 21, pp. 3675–3694, 1995.

[12] V. Schulze, F. Zanger, “Development of a simulation model to investigate tool wear in Ti-6Al-4V alloy machining,” in Advanced materials research, vol. 223, pp. 535–544, Trans Tech Publ, 2011.

[13] X. Xu, J. Zhang, H. Liu, Y. He, W. Zhao, “Grain refinement mechanism under high strain-rate deformation in machined surface during high speed machining ti6al4v,” Materials Science and Engineering: A, vol. 752, pp. 167–179, 2019.

[14] A. Mir, X. Luo, I. Llavori, A. Roy, D. L. Zlatanovic, S. N. Joshi, S. Goel, “Challenges and issues in continuum modelling of tribology, wear, cutting and other processes involving high-strain rate plastic deformation of metals,” Journal of the Mechanical Behavior of Biomedical Materials, vol. 130, 6 2022.

[15] F. Zanger, N. Boev, V. Schulze, “Novel approach for 3d simulation of a cutting process with adaptive remeshing technique,” Procedia CIRP, vol. 31, pp. 88–93, 2015. 15th CIRP Conference on Modelling of Machining Operations (15th CMMO).

[16] S. N. Melkote, R. Liu, P. Fernandez-Zelaia, T. Marusich, “A physically based constitutive model for simulation of segmented chip formation in orthogonal cutting of commercially pure titanium,” CIRP Annals, vol. 64, no. 1, pp. 65–68, 2015.

[17] G. Ortiz-de Zarate, A. Sela, A. Madariaga, T. Childs, P. Arrazola, “Sensitivity analysis of the input parameters of a physical based ductile failure model of Ti-6Al-4V for the prediction of surface integrity,” Procedia CIRP, vol. 87, pp. 533–538, 2020.

[18] A. Sela, D. Soler, G. Ortiz-de Zarate, G. Germain, F. Ducobu, P. J. Arrazola, “Inverse identification of the ductile failure law for ti6al4v based on orthogonal cutting experimental outcomes,” Metals, vol. 11, no. 8, p. 1154, 2021.

[19] D. Umbrello, A. Bordin, S. Imbrogno, S. Bruschi, “3d finite element modelling of surface modification in dry and cryogenic machining of ebm Ti6Al4V alloy,” CIRP Journal of Manufacturing Science and Technology, vol. 18, pp. 92–100, 2017.

[20] M. Saez-de Buruaga, L. Gainza, P. Aristimuno, D. Soler, G. Ortiz-de Zarate, O. Aizpuru, R. Mielgo, P. Arrazola, “Fem modeling of hard turning 42crmos4 steel,” Procedia CIRP, vol. 82, pp. 77–82, 2019.

[21] G. Ortiz-de Zarate, D. Soriano, A. Madariaga, A. Garay, I. Rodriguez, P. Arrazola, “Experimental and fem analysis of dry and cryogenic turning of hardened steel 100cr6 using cbn wiper tools,” Procedia CIRP, vol. 102, pp. 7–12, 2021.

[22] E. Segebade, M. Gerstenmeyer, F. Zanger, V. Schulze, “Cutting simulations using a commercially available 2d/3d fem software for forming,”Procedia CIRP, vol. 58, pp. 73–78, 2017. 16th CIRP Conference on Modelling of Machining Operations (16th CIRP CMMO).

[23] G. Ortiz-de Zarate, A. Sela, F. Ducobu, M. Saez-de Buruaga, D. Soler, T. Childs, P. Arrazola, “Evaluation of different flow stress laws coupled with a physical based ductile failure criterion for the modelling of the chip formation process of ti-6al-4v under broaching conditions,” Procedia CIRP, vol. 82, pp. 65–70, 2019.

[24] G. Ortiz-de Zarate, A. Madariaga, A. Garay, L. Azpitarte, I. Sacristan, M. Cuesta, P. Arrazola, “Experimental and fem analysis of surface integrity when broaching ti64,” Procedia Cirp, vol. 71, pp. 466–471, 2018.

[25] A. Sela, G. Ortiz-De-Zarate, D. Soler, G. Germain, L. Gallegos, P. J. Arrazola, “Adiabatic self-heating determination for ti6al4v at different temperatures,” International Journal of Heat and Mass Transfer, vol. 204, p. 123747, 2023.

[26] G. Ortiz-de Zarate, A. Madariaga, P. J. Arrazola, T. H. Childs, “A novel methodology to characterize tool-chip contact in metal cutting using partially restricted contact length tools,” CIRP Annals, vol. 70, no. 1, pp. 61– 64, 2021.