15th World Congress on Computational Mechanics (WCCM-XV)
8th Asian Pacific Congress on Computational Mechanics (APCOM-VIII)
Virtual Congress: 31 July – 5 August 2022
S. Koshizuka (Ed.)

MODELLING OF FORMING PROCESSES USING THE PARTICLE FINITE ELEMENT METHOD (PFEM)

JOSEP M. CARBONELL[1], HADI BAKHSHAN[2], JUAN M. RODRIGUEZ[3], AND EUGENIO OÑATE[4]

[1] MECAMAT group, Faculty of Science and Technology and Engineering
Universitat de Vic-Universitat Central de Catalunya (UVic-UCC)
Carrer de la Laura 13, 08500 Vic, Spain. e-mail: [1]cpuigbo@cimne.upc.edu

[3] Department of Mechanical Engineering, EAFIT University,
Medellin, Colombia, South America. e-mail: [3] jmrodrigup@eafit.edu.co

[2,4] Centre Internacional de Mètodes Numèrics en Enginyeria (CIMNE)
Universidad Politècnica de Catalunya (UPC)
Campus Nord UPC, 08034 Barcelona, Spain
e-mails: [2]hbakhshan@cimne.upc.edu, [4]onate@cimne.upc.edu

Key words: Forming processes, thermo-mechanical problems, particle methods, PFEM

Abstract

In this work we present the latest advances in the Particles Finite Element Method (PFEM) for the numerical modeling of forming processes. In the recent past, very good results of the method have been shown in the simulation of 3D cutting problems. The method has very good capabilities for treating large deformations in massive volumetric parts. Now the method is applied and extended to other forming operations: forging, blanking, minting, machining, etc., for metals and other materials.

One of the important aspects of these manufacturing techniques is the interaction with the molds and dies. Deformable contact interactions are needed to obtain a close correspondence between numerical and experimental results. The characterization of the thermomechanical interaction with the coatings of the tool plays an important role. Advances have been made in meshing techniques for treating three-dimensional parts and for the modeling deformable contact.

The characterization of friction and wear of the forming tools can be modeled considering also the lubrication on the surfaces. The purpose is to obtain the characteristics of the final shape of the workpiece, the areas that experience large plastic deformations and the residual stresses that remain in the processed material. This information is very valuable for the optimal design of the manufacturing operation.

To show the virtues of the method, several examples of forming operations are presented. The capabilities of the method are discussed, as well as the accuracy of the solutions.

1 INTRODUCTION

In the last decade the Particle Finite Element Method (PFEM) have gained considerable interest in the community solid mechanics as a method to tackle large displacement problems and changing boundaries. The origins of the method are in the field of computational fluid dynamics (CFD). It presented very attractive advantages in modelling fluids exhibiting moving free surfaces. These advantages come from the use of a Lagrangian description of the motion of the continuum medium combined with a continuous remeshing of the domain.

In this work, the Particle Finite Element Method is used for the simulation of metal forming. In solids, the Lagrangian approach is the usual way to deal with large deformations. However, numerical models use to fail when a mesh used in the discretization of the solid domain suffers from large distortions or large changes in the boundaries. To face this problem the particle description is useful, PFEM includes the use of a remeshing process, α-shape concepts for detecting domain boundaries and rebuilding the domain from a cloud of particles. This ensures domain discretization with well-shaped finite elements during the deformation process.

The governing equations for the deformable bodies are treated via a mixed formulation using simplicial elements with equal linear interpolation for displacements, pressure and temperature. Specific contact mechanics laws for the forming tools and material constitutive models for the workpieces are considered.

The merits of the formulation are demonstrated in the solution of 2D thermally coupled metal forming processes. The method shows good results and it is a promising method for future simulations of complex 3D forming operations. It also opens up new possibilities to predict highly non-linear problems and increase the accuracy of mechanical response computation.

2 FUNDAMENTALS OF THE PARTICLE FINITE ELEMENT METHOD

The particle finite element method emerged as a natural evolution of meshless methods [1-3]. The method includes a support mesh which is used for searches and calculations. It reduces the complexity of neighbor searches and improves the accuracy with non-smoothed results. In the present work, the PFEM follows next steps:

1. The starting domain is meshed and the nodes of the discretization are referred as the initial set of 'particles': The accuracy of the numerical solution is dependent on the considered number of particles; an alpha size is assigned to each area in order to achieve the desired number of particles in the next step.
2. Generate the finite element mesh using the previous set of particles: Previous mesh is used also to insert or release particles according the assigned size. Total cloud of points is reconnected by a Delaunay triangulation.
3. Identify the external boundaries to impose the boundary conditions and to compute the domain integrals. If a constrained Delaunay triangulation is used the boundary is preserved from the previous step.

4. Solve the non-linear Lagrangian form of the balance equations finding displacement, pressure and temperature.
5. Update the particle position using the computed values of displacements.
6. Go back to step 2 and repeat for the next time step.

In this solution flowchart, the numerical solution of the equations is the most critical part from the computational point of view. The generation of a new mesh and the identification of the boundaries is crucial but the computational cost is small compared with the solution of the equations. The Delaunay triangulation is adopted together with the so-called alpha shape method for boundary identification in the classical particle schemes. To keep high fidelity in the boundary shapes a constrained tessellation must be considered. If a constrained Delaunay tessellation is used the boundaries are conserved and the alpha shape technique is not needed anymore.

3 GOVERNING EQUATIONS FOR A LAGRANGIAN CONTINUUM

For the modelling of the metal pieces under large deformation conditions the Lagrangian description of the continuum is used. Particles are considered as material points, the domain (defined by this cloud of particles) occupies a certain current configuration at time t^n and reaches an updated configuration at time $t^{n+1} = t^n + \Delta t$. The volume V and its boundary Γ at the current and updated configurations are denoted as (V_n, Γ_n) and (V_{n+1}, Γ_{n+1}), respectively. The objective is to find the new updated configuration when computing each time step. Obtaining velocities, strain rates, stresses and temperatures at this point. Subsequently it becomes the current configuration for the next time step.

3.1 Momentum equations

The equation of conservation of linear momentum for a deformable continuum is written in an updated *Lagrangian* description as

$$\boldsymbol{\nabla} \cdot \boldsymbol{\sigma} + \mathbf{b} \ = \ \rho \mathbf{a} \quad \text{ in } V_{n+1} \times (0, t^{n+1}) \tag{1}$$

In Eq. (1) V_{n+1} is the analysis domain in the updated configuration at time t^{n+1} with boundary Γ_{n+1}, $\boldsymbol{a}$ and $\boldsymbol{b}$ are the acceleration and the body force components along the cartesian axis, ρ is the density, and $\boldsymbol{\sigma}$ are the Cauchy stresses in V_{n+1} .
For the mixed two-field form, having the displacement $\boldsymbol{u}$ and the volumetric deformation ϕ as independent variables (the $\boldsymbol{u} - \phi$ formulation) it is expressed as:

$$\left.\begin{aligned} \boldsymbol{\nabla} \cdot \hat{\boldsymbol{\sigma}} + \mathbf{b} \ &= \ \rho \mathbf{a} \quad \text{ in } V_{n+1} \times (0, t^{n+1}) \\ J - \phi \ &= \ \mathbf{0} \quad \text{ in } V_{n+1} \times (0, t^{n+1}) \end{aligned}\right\} \tag{2}$$

where J is the determinant of the deformation gradient $\mathbf{F} = \frac{d\mathbf{x}}{d\mathbf{X}}$, ϕ is the volumetric deformation and $\hat{\boldsymbol{\sigma}} = \boldsymbol{\sigma}(\hat{\mathbf{F}})$ is the Cauchy stress evaluated via the deformation gradient $\hat{\mathbf{F}}$, defined as

$$\hat{\mathbf{F}} = \left(\frac{\phi}{J}\right)^{\frac{1}{3}} \mathbf{F} \tag{3}$$

This formulation allows the consider the total definition of the stresses without splitting the deviatoric and volumetric part as occurs in the mixed $\boldsymbol{u} - p$ formulation [1], being p the scalar volumetric pressure.

To find out a stable solution for a mixed equal-order interpolation of the scalar and vector fields $\boldsymbol{\phi - u}$**,** the polynomial pressure projection (PPP) approach introduced by *Bochev* [5] is going to be used.

3.2 Thermal balance equation

The thermal balance equation in the updated configuration is written in a Lagrangian framework as

$$\boldsymbol{\nabla} \cdot (\kappa\, \boldsymbol{\nabla}\theta) + \mathbf{Q} \;=\; \rho c\, \dot{\theta} \quad\quad in\;\; V_{n+1} \times (0, t^{n+1}\,) \tag{4}$$

where θ is the temperature, c is the thermal capacity, κ is the thermal conductivity and $\mathbf{Q}$ is the heat source. Equations (1) and (4) are completed by the standard boundary conditions.

5 CONTACT MECHANICS

The particle finite element method uses the ancillary mesh created when meshing the convex hull subtracting the domain bodies. These elements are used to define the contact surface, the contact active set and to apply the contact constraint in the governing equations. The contact method used for the modelling of contact forces is the *contact domain* method [6]. The contact-domain ancillary mesh is generated through a constrained Delaunay tessellation of the exterior of the deformable domains -after a boundary shrinkage operation.

6 REPRESENTATIVE NUMERICAL SIMULATIONS

Next three forming processes (bending, roll forming and forging) are modelled using the PFEM. In order to illustrate the proposed methodology. In all cases a large strain thermo-elastoplastic constitutive model has been used to model the forming material.

6.1 Bending

This example (Figure 1) shows the bending of a hot metal plate at 800K. The upper tool is pushing the plate toward the lower die. In the figure, vectors depict the contact forces computed via the *contact domain* (mechanical contact), and colors depict the evolution of the temperature, dissipated by the plastic strain and transferred between the tools and the workpiece. The ancillary mesh does not appear in the figure but is used also to thermal contact between the metal domains.

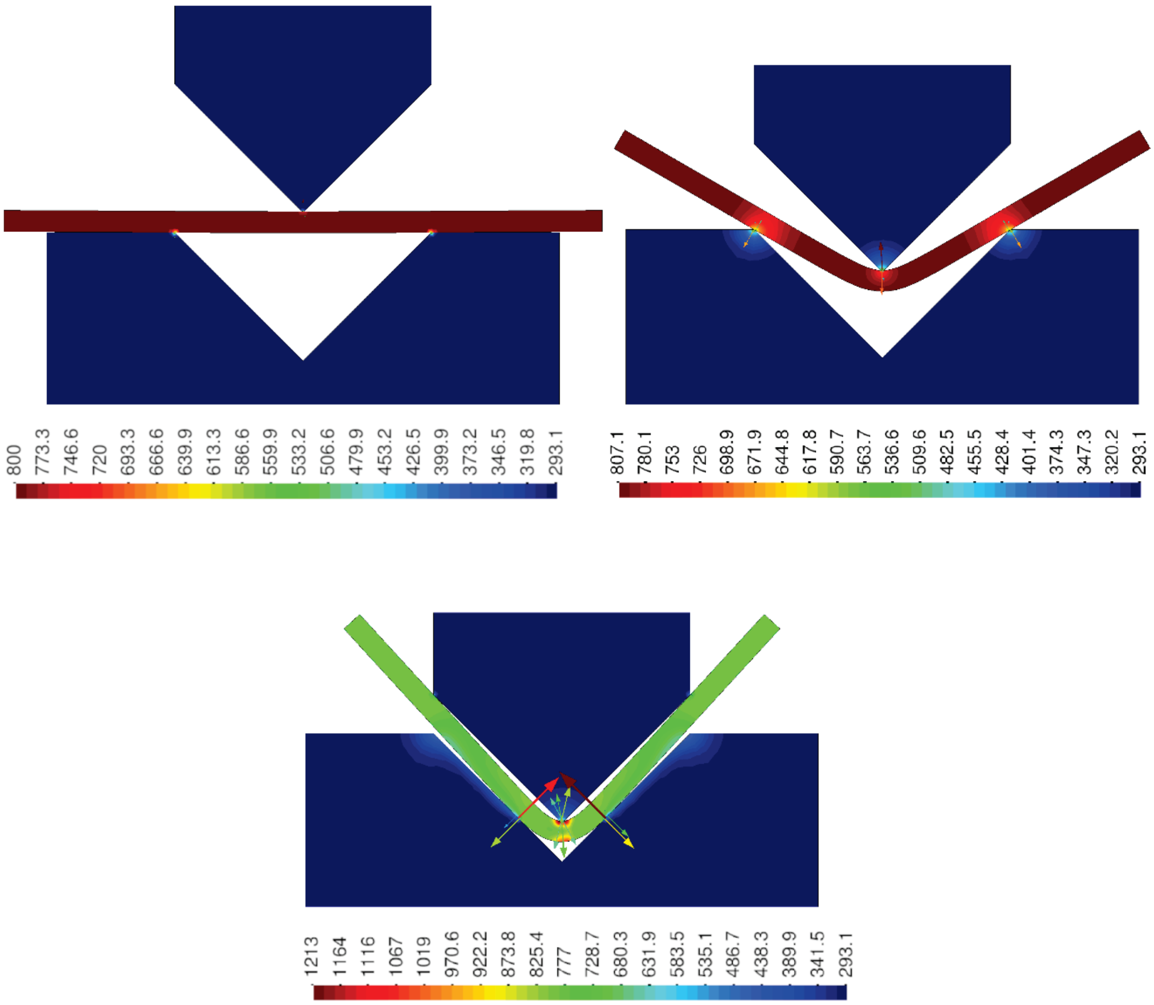

Figure 1: Hot bending of a steal plate. Representation of the contact forces and the temperatures in Kelvin.

6.2 Roll forming

In Figure 2 a roll forming example is presented. Two roll tools rotate at a constant speed and a metal plate approaches it until it is caught by the rolls. When the plate has been captured the friction with the rollers does the job of stretching and extruding the piece. In Figure 2 the mean pressure contour fill is depicted showing the compressed zones of the metal plate. Large strain thermo-elastoplastic model is considered and particles are inserted in the plastic zones to improve the accuracy of the results in these areas. This problem requires special attention in the conditions of friction and adhesion by contact because the roller tools are responsible for pulling the plate and extruding it, so if surfaces without friction are considered, the shaping is not carried out.

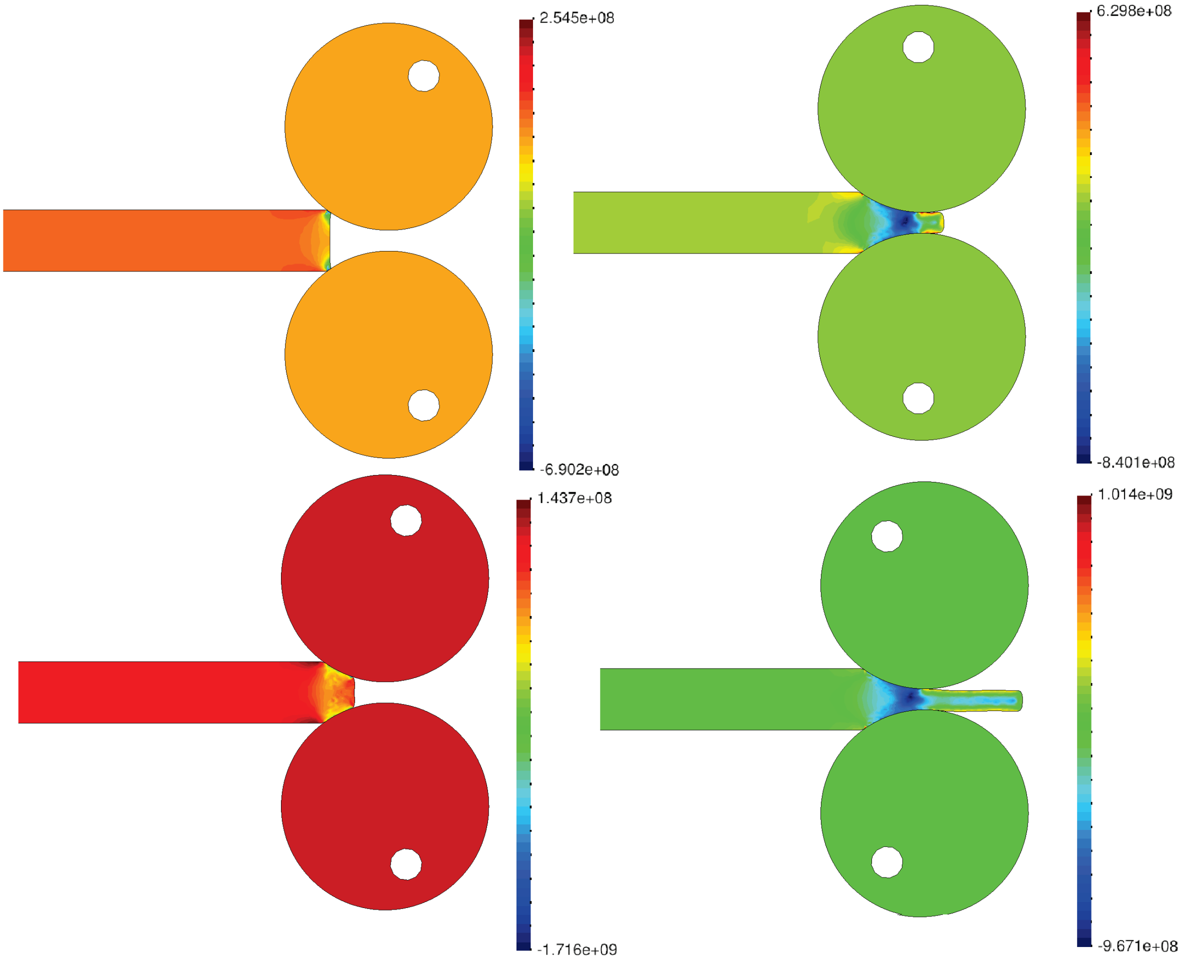

Figure 2: Roll forming example, two roll tools rotating at constant angular velocity and extruding a metal plate. Pressure contour fill depicted in pascals.

6.2 Forging

Last presented examples is a forging example, it reproduces the stamping of an axisymmetric cooper plate (a coin type plate). Figure 3 shows three instants of the process, at the beginning in the middle and at the end. The lower die was fixed at the base and the upper die was moving downwards. The results show the Von Mises stresses developed by the workpiece and the die. After analyzing the results, it is possible to detect areas with excessively high plastic deformation that will affect the final quality of the piece. The surface quality of the final product can be determined, as well as the areas where residual stresses remain. This information is very valuable from the point of view of the design of the forging process. As an example, in this particular case, the movements to achieve this shape with good quality must follow different forging operations.

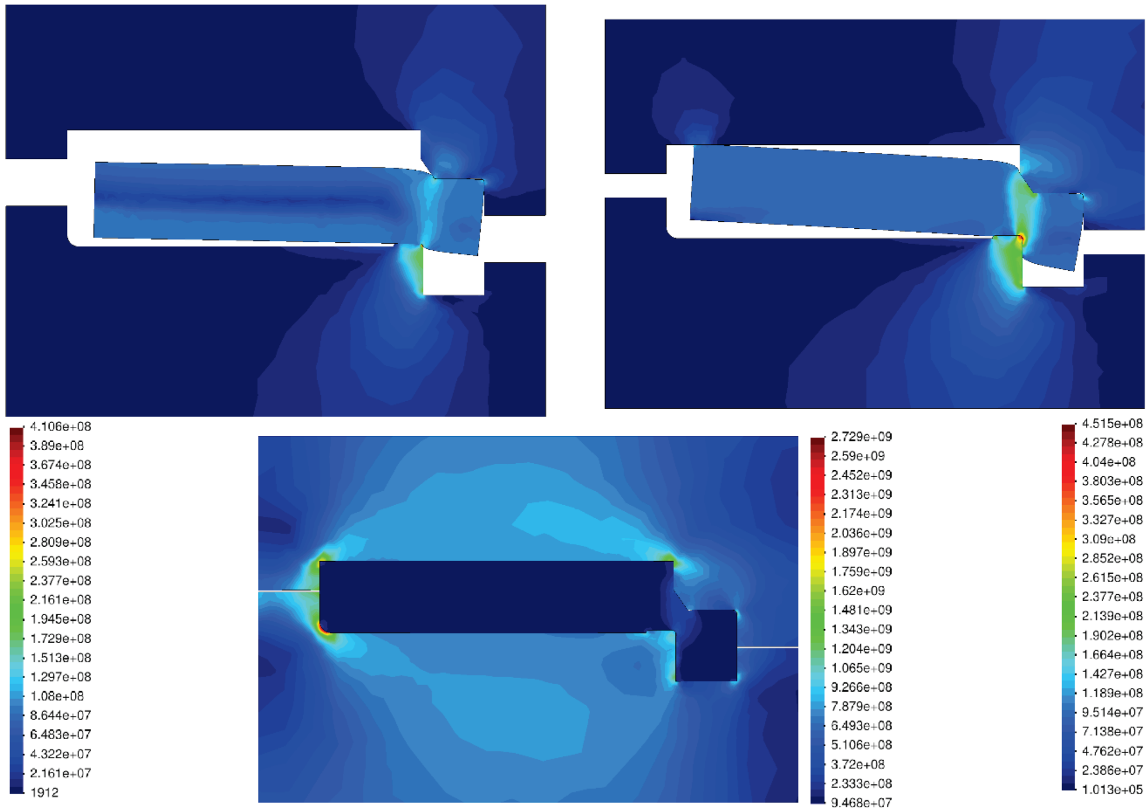

Figure 3: Forging example, circular metal plate forged between two deformable dies. Von Mises stress contour fill depicted in pascals.

5 CONCLUSIONS

A Lagrangian formulation is presented for the analysis of industrial forming processes involving thermally coupled interactions between deformable continua. The governing equations for the generalized continuum are discretized using elements with equal linear interpolation for the displacement and the temperature. The merits of the formulation in terms of its general applicability have been demonstrated in the solution of a variety of thermally-coupled industrial forming processes using the PFEM. Examples presented in this work are 2D plain strain and 2D axisymmetric, however there exist also the extension to 3D cases, see [1].
PFEM has gained much popularity in various fields of solid mechanics, machining, additive manufacturing [8], also very promising results are obtained when applied to penetration problems in geomechanics, see [9].

REFERENCES

[1] Carbonell, J.M.; Rodríguez, J.M.; Oñate, E.: Modelling 3D cutting problems with the Particle Finite Element Method. In: Computational Mechanics 66 (2020) No. 1, pp. 603-624.
[2] Rodríguez, J.M.; Carbonell, J.M.; Jonsén, P.: Numerical methods for the modelling of chip

formation processes. In: Archives of Computational Methods in Engineering 27 (2020) No. 1, pp 387-412.

[3] Oñate, E.; Franci, A.; Carbonell, J.M.: A particle finite element method for analysis of industrial forming processes. In: Computational Mechanics 54 (2014) No. 1, pp. 85-107.

[4] Oliver, J. C. et al.: Particle Finite Element Methods in Solid Mechanics Problems. In: Computational Methods in Applied Sciences 7 (2007) No. 1, pp. 87-103.

[5] Bochev, P. B.; Dohrmann, C. R.; Gunzburger, M. D.: Stabilization of Low-Order Mixed Finite Elements for the Stokes Equations. In: SIAM Journal on Numerical Analysis 44 (2008) No. 1, pp. 82-101.

[6] Hartmann, S., Oliver, J., Weyler, R. Cante, J. C., and Hernández, J.A. (2009). A contact domain method for large deformation frictional contact problems. Part 2: Numerical aspects. Computer Methods in Applied Mechanics and Engineering 198 (33–36), 2607-2631.

[7] J.M. Rodriguez, S. Larson, J.M. Carbonell, P. Jonsén (2021). Implicit or explicit time integration schemes in the PFEM modeling of metal cutting processes.

[8] B.-J. Bobach, R. Boman, D. Celentano, V.E. Terrapon, J.-P. Ponthot (2021). Simulation of the Marangoni Effect and Phase Change Using the Particle Finite Element Method. Appl. Sci. 2021, 11, 11893.

[9] J.M. Carbonell, Ll. Monforte, M. Ciantia, M. Arroyo and A. Gens (2022). Geotechnical particle finite element method for modeling of soil-structure interaction under large deformation conditions. Journal of Rock Mechanics and Geotechnical Engineering. Elsevier.