ISSN 2176-5480

22nd International Congress of Mechanical Engineering (COBEM 2013)
November 3-7, 2013, Ribeirão Preto, SP, Brazil

NUMERICAL MACHINING SIMULATION FOR AN AISI 304 STAINLESS STEEL CONSIDERING MICROFRACTURE MECHANICS ASPECTS

Gil Magno Portal Chagas
University of Sao Paulo USP – Polytechnic School - Av. Prof. Mello Moraes, 2231, 05508-900, São Paulo S.P.
Federal Institute of Santa Catarina - Campus Geraldo Werninghaus - Rua dos Imigrantes, 445, Vila Rau, Jaraguá do Sul S.C.
gilchagas@usp.br

Vanessa Seriacopi
University of Sao Paulo USP – Polytechnic School - Av. Prof. Mello Moraes, 2231, 05508-900, São Paulo S.P.
vanessaseriacopi@usp.br

Izabel Fernanda Machado
University of Sao Paulo USP – Polytechnic School - Av. Prof. Mello Moraes, 2231, 05508-900, São Paulo S.P.
machadoi@usp.br

Abstract. *Metal cutting is a complex process, since it comprises heat transfers, materials science and mechanical properties. The determination of the material behavior and the chip formation during the process has been a great challenge, due to the large plastic deformation, high strain rates, heat generation, and the tribological conditions involved. One of the theories to explain the chip formation problem takes into account the microfracture mechanics, in which ductile metals damage can be described in three steps: nucleation, growth, and coalescence of microvoids that initiate at points of stress concentration. This work describes the numerical machining simulations of an AISI 304 stainless steel by means of Finite Element Method. Orthogonal cutting was used to evaluate the mechanisms involved in the chip formation process. Two simulation groups were performed, one macro scale, analyzing the workpiece and tool, and the other one micro scale, representing a small area on the primary shear zone during chip formation. The complex stress fields in the primary shear zone, obtained in the macro simulation, were transferred to a small area, in which microvoids were added. This small region was simulated with the same macro scale conditions. The results showed the stress concentrations and the microvoids evolution during the simulation, and helps explaining the mechanisms involved in the process.*

Keywords: *Machining simulation, Finite Element simulation, Orthogonal cutting, Chip formation, Microvoids*

1. INTRODUCTION

There is a considerable amount of research in order to understand the complex phenomenon involved in metal cutting (Shi and Attia, 2010). This knowledge is essential to develop the cutting process improvement. Finite Element Method Simulation (FEM) has been successfully utilized in several works (Umbrello, 2008; Sima and Ozel, 2010; Pantel *et a*l., 2012) as a tool to numerically evaluate the metal cutting process. In applying FEM to metal cutting chip formation there are several important problems involved, among them:

1. an extremely high strains, and strain rates occurs. Besides, the severe deformation in the workpiece material at high temperatures occurs in a very small region, and makes it difficult to access experimentally. The extremely high strain rates involved are in the order of 10^3 to 10^6 s^{-1}, and the development of an analytical solution for the thermo visco plastic material behaviour that represents correctly the phenomenon is hard to be obtained. (Astakov, 2006)

2. and a new surface is being formed, in which the morphology of the chip is not known. There are different theories to explain the chip separation problem during machining. The FEM is used here to analyze the chip separation in terms of micro fracture mechanics. This is according to the theory that ductile metal damage occurs in three steps: nucleation, growth, and coalescence of microvoids that initiate at points of stress concentrations and leads to gross failure in shear. (Shaw, 2005)

This paper aims to obtain and evaluate the stress and strain fields that occur in the primary shear zone during chip formation, using FEM commercial codes, and considering a random distribution of defects in a ductile metal chip to evaluate the effect of stress concentrators in the cutting process, mainly in the chip separation.

To reach the objectives, two different FEM simulations groups were carried out. The first one analyzing the machining cutting process of the workpiece and tool, where the results showed the chip morphology, temperature distribution, and stress and strain fields in the chip and in the workpiece. The second simulation group considers a distribution of defects in the alloy, and was performed in a small area corresponding to a region inside the primary shear zone. The loading applied was equivalent to the obtained in the first simulation group, and the results showed the stress concentrations close to the defects that can lead to microcracks.

ISSN 2176-5480

2. MACHINING SIMULATION

2.1 Numerical procedures

This work describes the numerical machining simulations of an AISI 304 stainless steel by means of Finite Element Method. Orthogonal cutting was used to evaluate the mechanisms involved in the chip separation process.

Two simulations groups were performed, the first one, uses the commercial FEM software DEFORM-2D^{TM} to analyze the machining cutting process of the workpiece and tool. This software is optimized for the use in mechanical forming and machining, where large plastic deformation is involved. It uses a very efficient adaptive meshing technique that contributes to improve the quality of the results. Thus, the macro scale was tested, analyzing the machining cutting process of the workpiece and tool. Subsequently, the maximum and minimum normal stress in the primary shear zone, obtained in the macro simulation, were transferred to a small area (micro scale), representing the primary shear zone during chip formation.

Thus, the second simulation group was performed by means of the commercial FEM software Abaqus/Standard release 6.12 to simulate the micro scale, in which the small area described above and inside the primary shear zone. It was obtained the stress and strain, considering the presence of microvoids in the primary shear zone. The Abaqus is a more powerful tool to evaluate microstructural features than DEFORM-2D, since it is a general purpose finite element code that allows a better control over different parameters involved in the simulation, including the mesh, and the software output.

2.2 Numerical model of the macro machining cutting process with Deform

The commercial FEM software DEFORM-2D^{TM} was used to simulate the orthogonal cutting process of an AISI 304 austenitic stainless steel. An elastic-plastic, implicit, plane strain, Lagrangian, fully coupled thermo-mechanical model was developed for the simulation. According to Vaz et al. (2007) the Lagrangian formulation assumes that the mesh is attached to the material and follows its deformation . To prevent the mesh excessive element distortion a re-mesh procedure is used.

The input parameters in the FEM software are: material properties, machining parameters, geometry of workpiece and tool, constitutive law, number of elements and boundary condition, contact (friction) and damage model.

2.2.1 Machining macro simulation process

The thermal and material properties used were available on the software library. The workpiece length was taken with 6 mm and its height of 1.5 mm, and was meshed using initially 1772 isoparametrical quadrilateral elements. The cutting tool had 1.65 mm in height and length, clearance angle α=11°, rake angle γ_0=6°, and edge radius r= 50 µm, was modeled as rigid to reduce the calculation time, and meshed with 2127 elements. The cutting speed v_c was set to 100 m/min, and the feed rate f=0.2mm/mm. Figure 1 shows the initial configuration of the model and the workpiece and tool meshes.

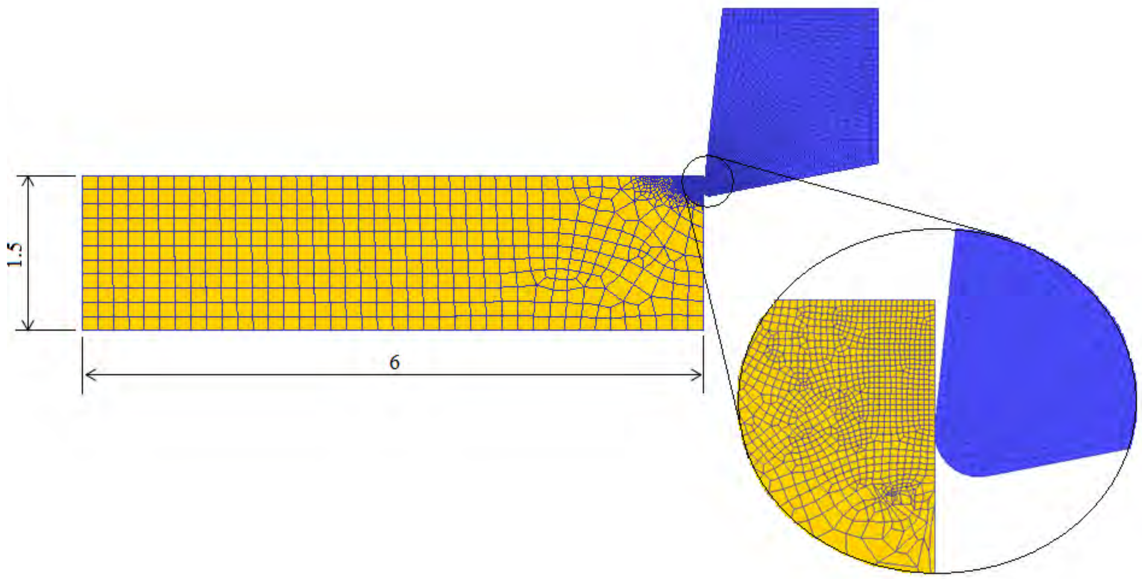

Figure 1. The initial model configuration and mesh.

ISSN 2176-5480

Constitutive law

It is necessary to apply an accurate model for the flow stress under high strain rates conditions. The Johnson and Cook (J.C.) constitutive model, established by Johnson and Cook (1983), is widely used in machining simulation. The model describes the flow stress of a material as the product of strain, strain rate and temperature effects, and it is utilized in this study. The J.C. constitutive model is given by:

$$\sigma = (A + B\varepsilon^n)\left(1 + C\ ln\frac{\dot{\varepsilon}}{\dot{\varepsilon}_0}\right)\left[1 - \left(\frac{T - T_r}{T_m - T_r}\right)^m\right] \tag{1}$$

where σ is the equivalent stress, ε is the equivalent plastic strain, $\dot{\varepsilon}$ is the strain rate, $\dot{\varepsilon}_0$ is the reference strain rate, A is the initial yield stress, B is the hardening modulus, C is the strain rate coefficient, n is the strain hardening coefficient, m is the thermal softening coefficient, T is the process temperature, T_r is the room temperature, and T_m is the melting temperature of the workpiece. The Johnson Cook parameters utilized are adopted from Lee et al. (2006), based on a Hopkinson impact bar experiment and presented in Tab. 1.

Table1. Johnson Cook parameters for AISI 304

A (MPa)	B (MPa)	C	n	m
310	1000	0.07	0.65	1

Tool-chip interface contact

The interfacial friction on the tool rake face is very complex, since the contact interaction on the interface workpiece-tool during machining occurs in two distinct zones: in the first sticking is the principal mechanism of friction and in the other sliding friction is the most important (Astakov, 2006; Ozel, 2006). However, due to the complex behavior a simple friction stress model was used, which is related to a global friction coefficient in this work and it is given by:

$$\tau = m_1 \tau_0$$

where m_1 is the friction constant, τ is the frictional stress, and τ_0 is the shearing flow stress. The friction constant is set to 0.4, according to previous studies (Agmel *et al.* 2011).

Damage criterion

The Cockroft and Latham damage criterion was selected in the software to predict the effect of stress on the chip segmentation during the orthogonal cutting, and the formation of the serrated chip morphology. The Cockroft and Latham's criterion is expressed as:

$$\int_0^{\varepsilon_f} \sigma d\varepsilon = C \tag{2}$$

where σ is the maximum principal stress, ε_f is the effective strain, and C the constant damage value. According to this criterion, when the integral of the largest tensile principal stress over the plastic strain path in Eq. 2 reaches the critical constant damage value C, fracture occurs and chip segmentation starts. (Umbrello, 2008)
The C constant was established according to a procedure adopted by Umbrello et al. (2007), Hua and Shivpuri (2004), and Chagas et al. (2013). The procedure consists in running several simulations with different values of C and comparing the results of the cutting force, feed force and chip thickness with the experiments. The constant C established in this simulation was C=150.

2.2.2 Model validation

To validate the model simulated with the software Deform, results of cutting force (Fc), feed force (F_f), and chip thickness (h') are compared with those obtained from experiments, and are presented in Tab. 2. The experimental results, with the same material and cutting parameters used in the simulation, were available in a previous work presented by Barbosa et al. (2013), the cutting and feed forces were measured with a three component piezoelectric turning dynamometer model 9265B/ 9441B, signal conditioner 5070A 11100 and a signal analyzer software DynoWare 2825A1-2 from Kistler.

Table 2. Numerical simulation (obtained in this work) and experimental Cutting forces, Feed forces and Chip thickness (Barbosa et al., 2013). The cutting speed (Vc) used to validate the numerical results was 100 m/min.

Cutting Speed Experimental and Simulated (m/min)	Experimental F_c (N)	Simulation F_c (N)	Experimental F_f (N)	Simulation F_f (N)	Experimental h'(mm)	Simulation h' (mm)
100	1052	1352	587	462	0.511	0.434

The difference between experimental and numeric results is due to several factors, including the constitutive model, whose Johnson Cook material constants utilized must be the same as those from the material workpiece. The cutting tool radius must be perfectly sharp in both experimental and numerical tests, though it cannot be guaranteed in the experimental tests. Finally, the simplifications of the model, and the complexities of the elasto visco plastic phenomenon involved, including the simple friction model utilized contributes to the difference. Considering the difficulties presented and an error of about 25% we can consider that the simulation showed a reasonable agreement with experiments. Besides, the results are in accordance with those obtained in other works such as Maranhão and Davim (2010).

2.2.4 Results from DEFORM-2D

The temperature distribution in the chip is shown in Fig. 2. The highest temperatures (of about 900^{o}C) occur in the secondary shear zone due to the high strain, and the friction in the tool and chip interface. The other region evaluated was the primary shear zone, where the temperatures obtained ranges between 440°C to 610°C.

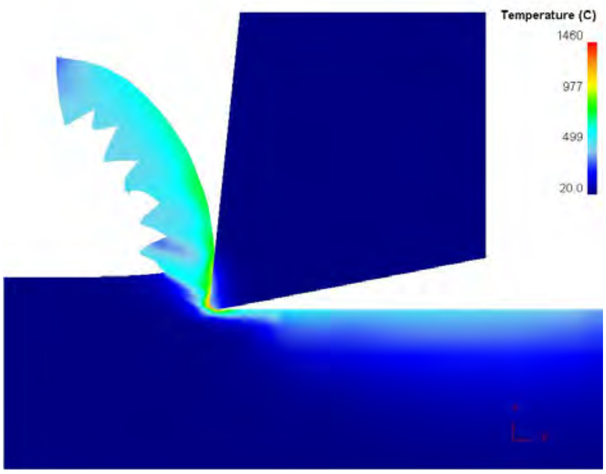

Figure 2. Temperature distribution in the chip

Figure 3 shows the sawtooth chip morphology, the similarities from experiment and simulation, and the effective strain obtained. It can be observed the non-homogenous morphology, regions with high strain along the shearing area, and regions less deformed within the chip segment.

ISSN 2176-5480

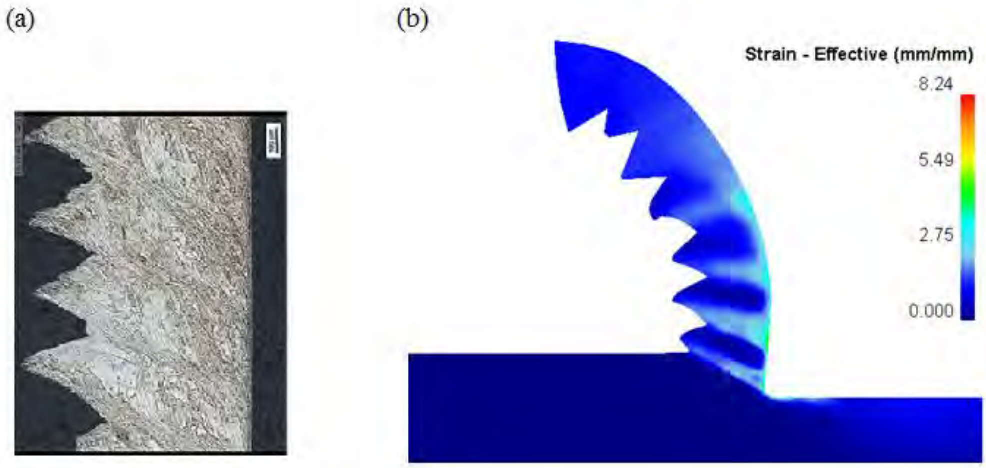

Figure 3. Chip Morphology and effective strain: (a) Experimental chip morphology of an AISI 304 machined (Barbosa et al., 2013), and (b) simulated chip morphology and the effective strain.

The maximum principal stress varies with a wide range along the primary shearing zone, as can be observed in Fig. 4. This variation can be due to the non-homogenous chip morphology obtained, and the damage model used, once there are regions inside the shearing zone with and without damage.

The maximum and minimum principal stresses obtained in one region, inside the primary shear zone, were selected to be utilized in the next simulation, where a micro region, represented by a rectangle in Fig. 4, is numerically evaluated using the software Abaqus.

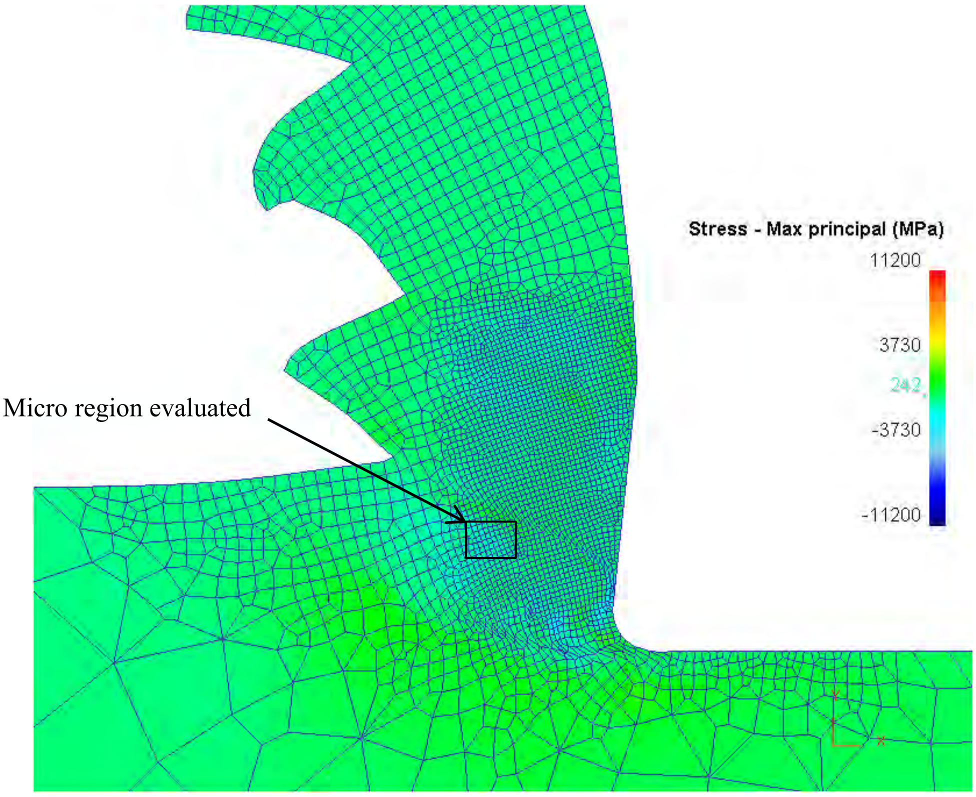

Figure 4. Maximum principal stresses in the chip with the FEM mesh.

ISSN 2176-5480

2.3 Numerical model of the micro region inside the primary shear zone with Abaqus

2.3.1 Numerical procedures

The commercial FEM software Abaqus /Standard was used to simulate the micro region of an AISI 304 austenitic stainless steel, which can be observed in a detail in Fig. 4, and correspond to the micro scale simulation. An elastic-plastic, implicit, plane strain, fully coupled thermo-mechanical model was developed for the simulation. Johnson Cook constitutive model was utilized. The simulation used the same materials, thermal parameters, and cutting speed from the previous macro scale machining simulation using DEFORM-2D. No damage model was utilized here.

Figure 5 shows the micro region workpiece dimensions, the boundary conditions, and the micro voids randomly distributed. The microvoids also displays different sizes, distribution, positions and orientations which will be related to the applied loads. The loads applied are pressures equivalent to the maximum and minimum normal stresses, and were obtained from the previous macro machining simulation in the primary shear zone. The loads were applied during an interval of time of $9x10^{-5}$s, equivalent to the cutting speed.

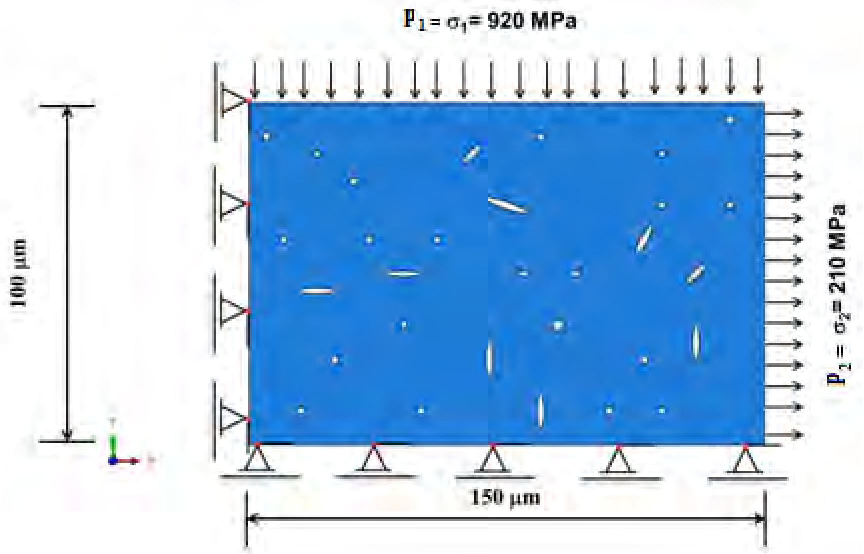

Figure 5. Workpiece dimensions with the loading and boundary conditions.

The workpiece was modeled with 34508 quadrilateral and triangular isoparametrical elements type CPE4RT and CPE3T. The mesh showing the initial micro voids is presented in Fig.6.

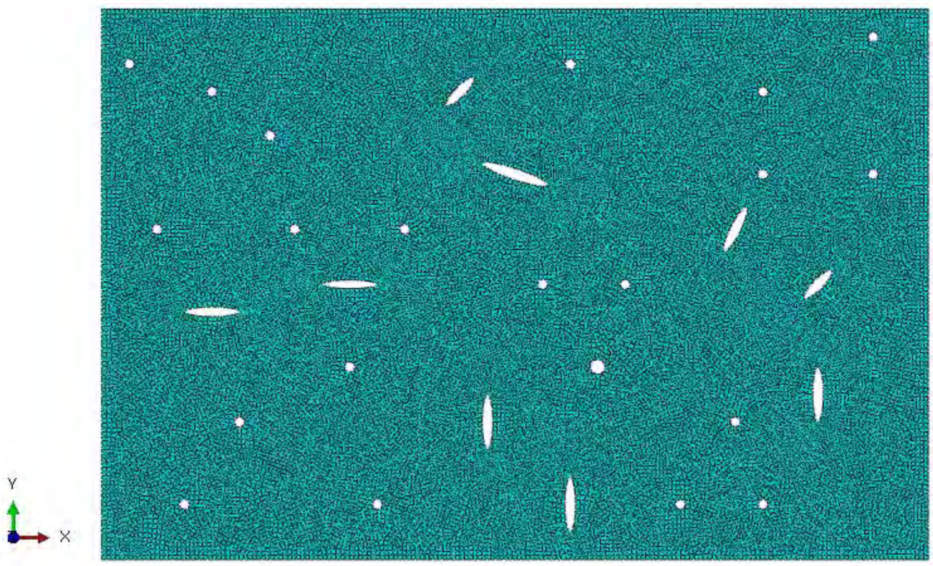

Figure 6. Mesh and the initial voids.

ISSN 2176-5480

2.3.2 Results from Abaqus

In order to evaluate the stresses and strains evolution, no damage model was utilized in the simulation. Hence, this results in no cracks initiation and propagation.

Figure 7 shows the stress concentration near the micro voids, and its evolution from the time of 2.5×10^{-9} s (Fig.7a) to 9×10^{-5} s (Fig.7c). It can also be observed that the stresses near the voids are about 3 to 4 times higher than the matrix stress. There is a change in the morphology of voids, which seems to be related to orientation between the voids and the loads applied (cutting forces). Figure 7 also shows an increase in the stress concentration near the voids, where cracks can initiate due to the large strain (Fig. 8). In the final force application time, according to Fig.7c, some voids are closed due to the high compression force, and there is a high stress concentration between one void and the next one. This can lead to a path where a crack can propagate as discussed by Shaw (2005). As a final comment the voids main features and the loads applied can be related.

(a)

(b)

(c)

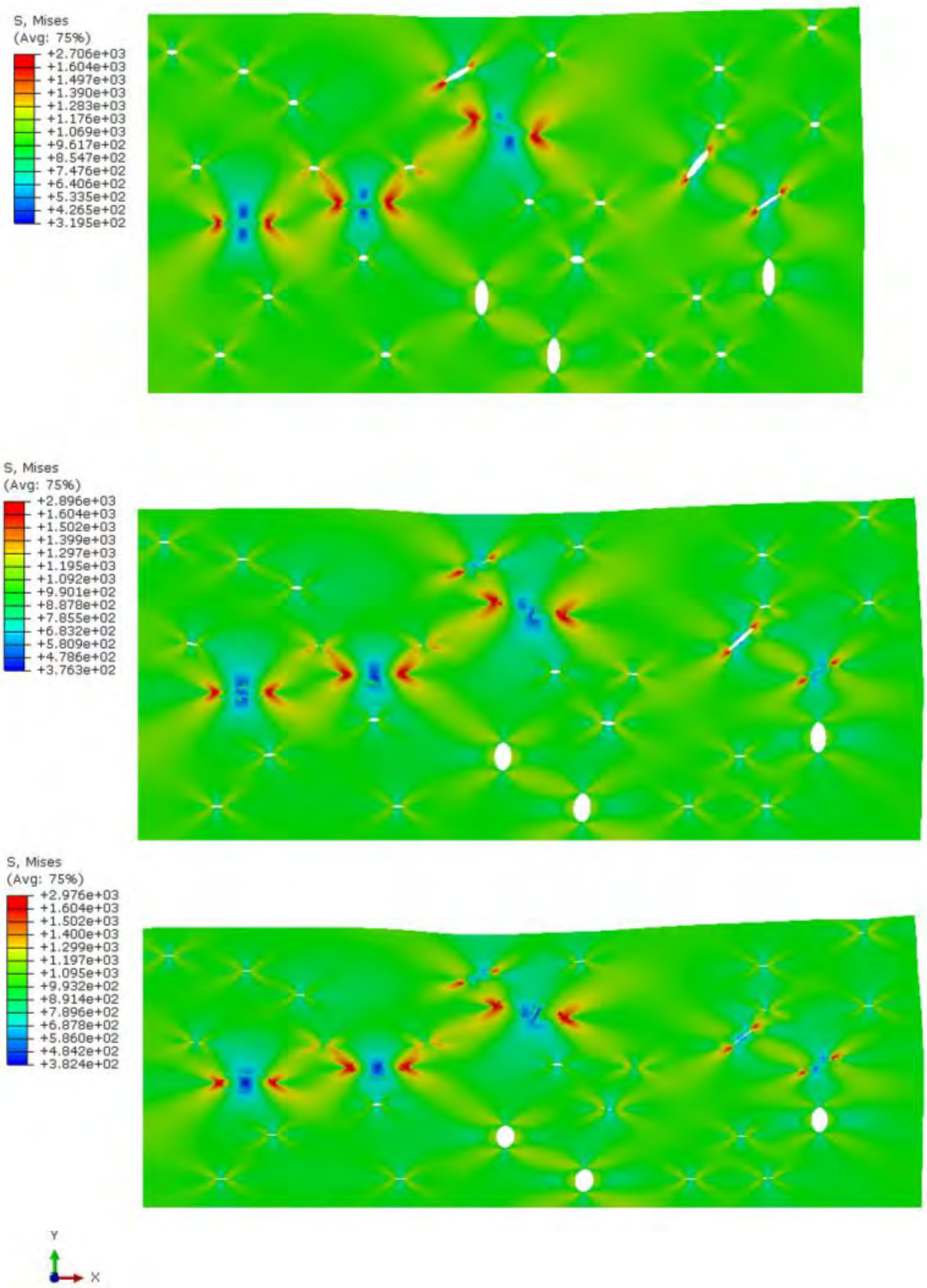

Figure 7. Stress behavior evolution: (a) Von Mises Stress in the instant time of 2.5×10^{-9}s, (b) Von Mises Stress in the instant time of 4.35×10^{-7}s, (c) Von Mises Stress in the final force application time of 9×10^{-5}s.

Figure 8 shows the equivalent plastic strain behavior evolution, from the time of 3x10^{-6}s (Fig. 8a), to the time of 9x10^{-5}s (Fig.8c). It can be observed an increase in the deformation with the force application time. The high localized strain (above yield strain, plastic deformation) near the voids confirms the regions where the cracks can initiate, and the regions between the voids, in green, where the cracks can propagate as evaluated due to stress conditions according to Fig 7.

High compressive forces applied in the y direction caused a reduction of 38 μm in height, and the tensile forces in the x direction caused an increase of 54 μm in width. The geometry changes are mainly due to the material non homogeneity.

(a)

(b)

(c)

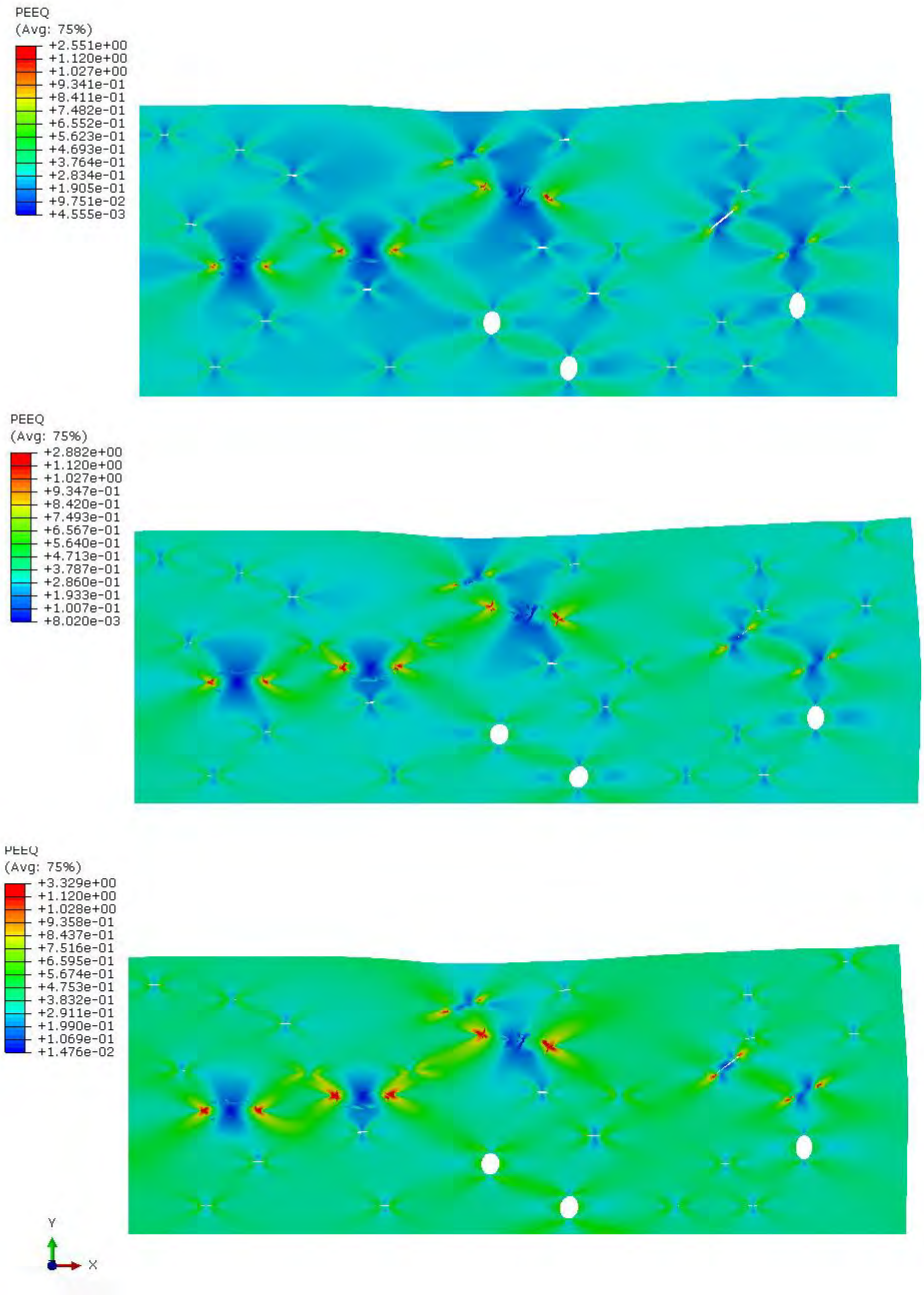

Figure 8. Plastic strain behavior evolution: (a) in the instant time of 3x10^{-6}s, (b) instant of time of 1.5x10^{-5}s, (c) final instant of time of 9x10^{-5}s.

3. FINAL REMARKS

This work allows the following evaluation, which is preliminary:

1. The macro simulation of cutting process by means of DEFORM-2D was considered validated as the error was about 25% and in accordance to the literature.
2. The results of micro scale simulation were used to show the effect of voids presence as well as its distribution, morphology and orientation between voids and applied loads.

It is worth mentioning, the existence of microvoids in a steel increase the stress concentration and can lead to a crack initiation and propagation. This mechanism can make it easier to machine the metal, comparing with a metal with absence of microvoids. For instance, the inclusions in austenitic stainless steels, such as manganese sulfide (MnS), can lead to a void nucleation, and growth, similar to the mechanism presented. Thus, the FEM has shown that it is a tool available to apply numerical model to investigate the complex mechanisms involved in the metal cutting machining process.

4. ACKNOWLEDGEMENTS

The authors are grateful to CNPq and FAPESP, for their financial support.

5. REFERENCES

Agmel, M., Ahadi,A., Stahl,J.E., 2011. “ A fully coupled thermomechanical two-dimensional simulation model for orthogonal cutting: formulation and simulation”. J. Engineering manufacturing, proc. IMechE, vol. 225 PartB.

Astakov, V.P., 2006. *Tribology of metal cutting.* Elsevier Science, 1ed, 292p.

Barbosa, P.A., Barbosa, C.A. Machado, I.F., 2013, “Analysis of machining parameters on the stresses into primary shear”, 7th Brazilian Congress on manufacturing engineering. (In Portuguese)

Chagas, G.M.C., Barbosa, P.A., Barbosa, C.A., Machado, I.F., 2013. “ Thermal analysis of the chip formation in austenitic stainless steel”. Procedia CIRP, vol. 8, p.293-298.

Hua, J. Shivpuri, R., 2004, “Prediction of chip morphology and segmentation during the machine of titanium alloys”, Journal of materials processing technology, vol. 150, p.124-133.

Johnson, G. R.; Cook, W. H., 1983. “A constitutive model and data for metals subjected to various strains, high strain rates, and high temperatures”. Proceedings of the Seventh International Symposium on Ballistics, p.541-547.

Lee, S., Barthelat, F., Hutchinson, J.W., Espinosa, H.D., 2006. “Dynamic failure of metallic pyramidal truss core materials - Experiments and modeling” . Int.J. Plasticity, vol. 22, p. 2118–2145.

Maranhão, C., Davim, J.P., 2010. “Finite element modeling of machining of AISI 316 steel: Numerical simulation and experimental validation”. Simulation Modeling Practice and Theory, vol. 18, p. 139-15.

Pantel, A.M., Fontaine,M., Thibaud, S., Gelin,J.C., 2012. “3D FEM simulations of shoulder milling operations on a 304L stainless steel”. Simulation Modelling Practice and Theory, vol. 22, p. 13-27.

Sima, M., Ozel, T. 2010. “Modified material constitutive models for serrated chip formation simulations and experimental validation in machining of titanium alloy Ti–6Al–4V”. International Journal of Machine Tools & Manufacture, vol. 50, p. 943-960.

Ozel, T., 2006. “The influence of friction models on finite element simulations of machining”. International Journal of Machine Tools & Manufacture, vol. 46 p.518–530.

Umbrello, D., M´Saoubi, R., Outeiro, J.C., 2007, “The influence of Johnson-Cook material constants on finite element simulation of machining of AISI 316L steel”. International Journal of Machine Tools & Manufacture, 47, 462-470.

Umbrello, D., 2008. “Finite element simulation of conventional and high speed machining of Ti6Al4V alloy”. Journal of materials processing technology,vol.196, p.79-87.

Shaw, M. C., 2005. *Metal cutting principles*. New York: Oxford University Press, 2 ed., 651p.

Shi B., Attia H. , 2010. “Current status and future direction in the numerical modeling and simulation of machining processes: a critical literature review”. Machining Science and Technology, 14:2, p.149-188.

Vaz JR, M., Owen, D.R.J., Kalhori, V., Lundblad, M.,2007. “Modelling and simulation of machining processes”. Arch Comput Methods Eng., v.14, p.173-204.

6. RESPONSIBILITY NOTICE