Rapid estimation of notch stress intensity factors in 3D large-scale welded structures using the peak stress method

Alberto Campagnolo and Giovanni Meneghetti

1 University of Padova, Department of Industrial Engineering, 35131 Padova, Italy

Abstract. The Peak Stress Method (PSM) is an engineering, FE-oriented application of the notch stress intensity factor (NSIF) approach to fatigue design of welded joints, which takes advantage of the singular linear elastic peak stresses from FE analyses with coarse meshes. Originally, the PSM was calibrated to rapidly estimate the NSIFs by using 3D, eight-node brick elements, taking advantage of the submodeling technique. 3D modelling of large-scale structures is increasingly adopted in industrial applications, thanks to the growing spread of high-performance computing (HPC). Based on this trend, the application of PSM by means of 3D models should possibly be even more speeded up. To do this, in the present contribution the PSM has been calibrated under mode I, II and III loadings by using ten-node tetra elements, which are able to directly discretize complex 3D geometries without the need for submodels. The calibration of the PSM has been carried out by analysing several 3D mode I, II and III problems. Afterwards, an applicative example has been considered, which is relevant to a large-scale steel welded structure, having overall size on the order of meters. Two 3D FE models, having global size of tetra elements equal to 5 and 1.66 mm, have been solved by taking advantage of HPC, being the global number of degrees of freedom equal to 10 and 140 millions, respectively. The NSIFs values estimated at the toe and root sides according to the PSM have been compared with those calculated by adopting a shell-to-solid technique.

1 Introduction

On the basis of the fundamental contributions of Williams [1], who studied two-dimensional notch problems under mode I (opening) and mode II (sliding) loadings, and Qian and Hasebe [2], who analysed the notch problem under mode III (tearing) loading, the singular linear elastic stress distributions in the neighborhood of a sharp V-notch tip, see the example of a toe side in a welded joint in Fig. 1, can be written as functions of the notch stress intensity factors (NSIFs), which quantify the intensity of the asymptotic stress fields. The following equations define the mode I, II and III NSIFs, respectively, according to Gross and Mendelson [3]:

\[ K_1 = \sqrt{2\pi} \lim_{r \to 0} \left( \sigma_{\theta\theta} \right)_{r=0} \cdot r^{1-\lambda_1} \]  
\[ K_2 = \sqrt{2\pi} \lim_{r \to 0} \left( \tau_{\theta r} \right)_{r=0} \cdot r^{1-\lambda_2} \]  
\[ K_3 = \sqrt{2\pi} \lim_{r \to 0} \left( \tau_{\theta z} \right)_{r=0} \cdot r^{1-\lambda_3} \]

(1a)(1b)(1c)

In Eqs. (1a)-(1c), \( \lambda_1, \lambda_2 \) and \( \lambda_3 \) are the mode I, II and III eigenvalues, respectively, which are dependent on the opening angle \( 2\alpha \) [1,2], while \( \sigma_{\theta\theta}, \tau_{\theta r} \) and \( \tau_{\theta z} \) represent the stress components evaluated along the notch bisector line, which is defined by the angular coordinate \( \theta = 0 \) according to Fig. 1.
In the literature, the NSIF approach has been used to analyse the medium as well as the high-cycle fatigue strength of specimens weakened by sharp V-notches and made of structural materials [4,5]. Focusing on welded joints, NSIFs allow to correlate the fatigue strength under uniaxial [6–9] as well as multiaxial loadings [10].

Nevertheless, it should be noted that the calculation of NSIFs on the basis of the results of numerical analyses shows a major drawback in engineering applications, since very refined FE meshes (element size on the order of $10^{-5}$ mm) are required in order to apply definitions (1a)-(1c) to calculate the NSIFs. When dealing with three-dimensional notched components, both the solution of the FE model and the post-processing of numerical results could be even more time-consuming.

In order to overcome this issue, an engineering and rapid technique, the so-called Peak Stress Method (PSM), has been proposed, which allows to speed up the calculation of the NSIFs by adopting coarse FE analyses, the element size being some orders of magnitude larger than that required to apply definitions (1a)-(1c). The second advantage of the PSM is that only a single stress value is necessary to estimate the NSIFs, instead of a number of stress-distance results, as usually necessary in order to apply definitions (1a)-(1c). The method takes inspiration from the contribution by Nisitani and Teranishi [11,12], who proposed a technique to readily estimate the mode I Stress Intensity Factor of a crack propagating from an ellipsoidal cavity. The PSM has been theoretically justified and extended to allow the rapid calculation also of the NSIF relevant to sharp and open V-notches under mode I [13,14], the SIF of cracks under mode II [15] and, finally, the NSIF of open V-notches under mode III [16]. It is worth noting that any NSIF-based approach for the structural strength assessment can in principle be reformulated on the basis of the PSM. Recently, the PSM has been applied in combination with the approach based on the averaged strain energy density (SED) to assess the fatigue strength of welded joints under axial [15,17], torsion [16] and multiaxial [18,19] loading conditions. Practically, the NSIFs $K_1$, $K_2$ and $K_3$ can be readily estimated according to the PSM by adopting the singular, linear elastic, opening (mode I), sliding (mode II) and tearing (mode III) peak stresses $\sigma_{0,0,0,\text{peak}}$, $\tau_{0,0,0,\text{peak}}$ and $\sigma_{0,0,0,\text{peak}}$, respectively, which are referred to the V-notch bisector line, according to Fig. 2, and calculated at the V-notch tip from FE analyses with coarse meshes. The estimated NSIF values can be obtained from the following expressions [13,15,16]:

$$K_1 \approx K_{FE}^{**} \cdot \sigma_{0,0,0,\text{peak}} \cdot d^{\frac{1}{2}}$$  \hspace{1cm} (2a)

$$K_2 \approx K_{FE}^{**} \cdot \tau_{0,0,0,\text{peak}} \cdot d^{0.5}$$  \hspace{1cm} (2b)

$$K_3 \approx K_{FE}^{**} \cdot \tau_{0,0,0,\text{peak}} \cdot d^{\frac{1}{2}}$$  \hspace{1cm} (2c)

where $a$ is the characteristic size of the considered sharp V-notch, as an example it is the notch depth in next Fig. 4b, $d$ is the so-called ‘global element size’, i.e. the average FE size adopted by the free mesh generation algorithm available in the numerical software.

Parameters $K_{FE}^{**}$, $K_{FE}^{***}$ and $K_{FE}^{****}$ depend on the calibration options: (i) element type and formulation; (ii) mesh pattern of finite elements and (iii) procedure for stress extrapolation at FE nodes. See the detailed discussion reported in [20].

Originally, 2D, four-node plane quadrilateral elements of Ansys® element library [13,15,16] were adopted to calibrate parameters $K_{FE}^{**}$, $K_{FE}^{***}$ and $K_{FE}^{****}$ which resulted equal to 1.38, 3.38 and 1.93, respectively, such values being valid under the conditions discussed in the relevant literature [13–16], to which the reader is referred. Recently, $K_{FE}^{**}$ and $K_{FE}^{***}$ have been also calibrated by adopting six commercial FE packages other than Ansys®, taking advantage of a Round Robin between Italian Universities [20].

Afterwards, the PSM has been extended to be used with 3D, eight-node brick elements [14], taking advantage of the submodeling technique available in Ansys® software. More precisely, when dealing with complex 3D joint geometries a submodel consisting of brick elements was generated after having analysed a main model meshed by employing ten-node tetra elements.

Three-dimensional modelling of large-scale, or even full-scale, structures is increasingly adopted in industrial applications, thanks to the growing spread of high-performance computing (HPC). Based on this trend, the
2 Calibrating the PSM with 10-node tetrahedral elements

When adopting tetrahedral elements to analyse a 3D notch problem, the mesh pattern obtained by the free mesh generation algorithm is intrinsically not regular, so that a node belonging to the notch tip could be shared by a different number of elements having significantly different shape. Therefore, the peak stress could vary along the notch tip profile even in the case of a constant applied NSIF. Figure 3 shows an example of the variability of the peak stress along the notch tip profile of a plate subjected to pure tension loading (see next Fig. 4b) and analysed by adopting a 3D FE model in which a plain strain condition has been simulated, as will be discussed in more detail in the following. For comparison purposes, the reciprocal of the normalised number of elements (\(N^\circ\) FE/max \(N^\circ\) FE)\(^1\), which share the node at which peak stress is evaluated, has been reported in Fig. 3. It can be observed that the higher the number of elements sharing the node, the lower the peak stress.

![Graph](image)

**Fig. 3.** Example of variability of the peak stress \(\sigma_{\theta \theta, 0, \text{peak}}\) calculated at vertex nodes belonging to the V-notch tip profile and comparison with the number of elements sharing each node (\(N^\circ\) FE). Definition of the average peak stress \(\bar{\sigma}_{\theta \theta, 0, \text{peak}}\).

Accordingly, to reduce the variability of the peak stress along the notch tip profile, an average peak stress value has been introduced (see red markers in Fig. 3), being defined at the generic node \(n=\)k as the moving average on three adjacent vertex nodes, i.e. \(n=k-1, k\) and \(k+1\):

\[
\bar{\sigma}_{\theta \theta, 0, \text{peak}, n} = \frac{\sigma_{\theta \theta, 0, \text{peak}, n-k-1} + \sigma_{\theta \theta, 0, \text{peak}, n-k} + \sigma_{\theta \theta, 0, \text{peak}, n-k+1}}{3}
\]

It is worth noting that only peak stress values calculated at vertex nodes of the quadratic tetrahedral elements have to be input in Eq. (3), i.e. stress values at mid-side nodes located at the notch tip profile, which are provided by Ansys® when “path operations” or “GET commands” are employed in post-processing, must be neglected. On the other hand, when “list nodal results” or “query results” are adopted in the post-processing environment of Ansys® code, stress values at mid-side nodes are automatically excluded.

Given this, the PSM has been calibrated by analysing several 3D notch problems under pure mode I, pure mode II and pure mode III loadings. After having calculated the peak stresses, the parameters \(K^FE\), \(K^\circ\) and \(K^\circ\) have been evaluated from Eqs. (2a)-(2c), re-arranged as follows:

\[
K^FE = \frac{K_1}{\sigma_{\theta \theta, 0, \text{peak}} d^{1.5}}
\]

\[
K^\circ = \frac{K_2}{\tau_{\theta \theta, 0, \text{peak}} d^{0.5}}
\]

\[
K^\circ = \frac{K_3}{\tau_{\theta \theta, 0, \text{peak}} d^{-1.5}}
\]

The same material properties have been adopted in all FE analyses, consisting in a structural steel, having Young’s modulus equal to 206000 MPa and a Poisson’s ratio \(\nu\) of 0.3.

3D, ten-node, quadratic tetrahedral elements (SOLID 187 of Ansys® element library) have been used in all FE analyses. The numerical integration has been carried out by adopting 4 Gauss points, i.e. a reduced integration, being the sole element formulation available in Ansys®. Once selected the proper element type, the average FE size \(d\) has been the sole parameter used to drive the automatic free mesh generation algorithm.

Dealing with mode I and mode II problems, in order to obtain a uniform distribution of the relevant NSIFs along the notch tip profile, either plane-strain or plane-stress analyses have been performed:

- plane strain state was simulated by constraining the out-of-plane element displacement \(u_z\) (and thus the corresponding \(\epsilon_z\) strain component, \(\epsilon_z = 0\));
- plane stress state was simulated by defining an ideal orthotropic material having out-of-plane \(s_z\) strain uncoupled from the in-plane components, i.e. \(\epsilon_x\) and \(\epsilon_y\). For this aim, \(v_{xy}\) and \(v_{yz}\) Poisson coefficients were set to zero. In absence of coupling of in-plane with out-of-plane strains, being \(\epsilon_z = 0\), then \(\sigma_{zz} = 0\), i.e. plane stress conditions are applied.
2.1. 3D problems (plane strain/plane stress), mode I loading, \(0^\circ < 2\alpha < 135^\circ\)

Several three-dimensional notch and crack problems under pure mode I (see Fig. 4 and Table 1) have been analysed. The considered case studies are a selection of those adopted in the first calibration of the PSM under mode I loading, which had been carried out by considering 2D [13] and 3D [14] problems. Here, those case studies have been extruded to obtain 3D components having thickness equal to \(s\).

Table 1. Geometrical and FE parameters considered in the calibration of the PSM with 10-node tetrahedral element under mode I, mode II and mode III loadings.

<table>
<thead>
<tr>
<th>Figure</th>
<th>Geometrical parameters</th>
<th>Mesh size (d) [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>4a</td>
<td>(2a = 3 \div 25) mm</td>
<td>(0.5 \div 1) mm</td>
</tr>
<tr>
<td>(b = 3 \div 24) mm (c = 2.2 \div 20.5) mm (s = 80) mm</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4b</td>
<td>(a = 1 \div 20) mm</td>
<td>(0.5 \div 10) mm</td>
</tr>
<tr>
<td>(2\alpha = 0, 90, 135^\circ) (w = 50) mm (s = 80) mm</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>(2a = 2 \div 200) mm</td>
<td>(0.5 \div 5) mm</td>
</tr>
<tr>
<td>(w = 400) mm (s = 40, 80, 160) mm</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>(z = 6.3 \div 9) mm</td>
<td>(1 \div 10) mm</td>
</tr>
<tr>
<td>(t = 7 \div 10) mm</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

3D linear elastic analyses have been performed by simulating plane strain or plane stress conditions and by adopting ten-node, quadratic tetrahedral elements, in order to calculate the peak stress values. Only one eighth of each geometry has been modeled by using the triple symmetry condition. The free mesh generation algorithm available in Ansys code has been used after having input the average element size \(d\). The mesh density ratio \(a/d\) has been varied between 1 and 20, by varying either the characteristic size of the notch problem, i.e. \(a\), or the element size \(d\) (see Table 1). Moreover, three values of the notch opening angle, i.e. \(2\alpha = 0^\circ, 90^\circ, 135^\circ\) have been considered, \(0^\circ\) and \(135^\circ\) being typical for weld root and toe side, respectively. A nominal gross-section stress equal to 1 MPa has been applied to each FE model.

After having solved the FE model, the peak value of the opening stress \(\sigma_{11,\text{peak}}\), being almost equal to \(\sigma_{\theta\theta,\theta=0,\text{peak}}\) in all cases of Figs. 4a and 4b, has been calculated at vertex nodes belonging to the V-notch tip profile, then Eq. (3) has been applied to derive the average peak stress at each vertex node.

2.2 3D problems (plane strain/plane stress), mode II loading, \(2\alpha = 0^\circ\)

A plate having the geometry shown in Fig. 5 (see also Table 1), weakened by a central crack \((2\alpha = 0^\circ)\) and subjected to pure mode II loading has been analysed. The case study has been taken from the first calibration of the PSM under mode II loading for 2D problems [15] and it has been extruded to obtain a 3D plate having thickness equal to \(s\).

3D linear elastic analyses have been carried out by simulating plane strain or plane stress conditions and by adopting ten-node, quadratic tetrahedral elements, in order to calculate the peak stress values. The mesh density ratio \(a/d\) has been varied in a wide range from 1 to 200, being \(2 < 2\alpha < 200\) mm. Only one eight of the plate geometry has been modeled by using the double anti-symmetry on planes \(Y-Z\) and \(X-Z\) and the symmetry on plane \(X-Y\).

The external load has been applied to the model by means of displacements \(u_x = u_y = 1.262 \cdot 10^{-3}\) mm at the plate free edges, which correspond to a nominal gross shear stress of 1 MPa when no crack is present. After having solved the numerical model, the peak value of the (mode II) shear stress \(\tau_{\theta=0,\text{peak}} = \tau_{xy,\text{peak}}\) has been calculated at the vertex nodes belonging to the crack tip profile, then Eq. (3) has been applied to derive the average peak stress at each vertex node.
2.3 3D problems, mode III loading, $2\alpha=0^\circ$, 135°

The weld root ($2\alpha = 0^\circ$) as well as the weld toe ($2\alpha = 135^\circ$) sides of tube-to-flange joints subjected to pure mode III loading, see Table 1 and an example in Fig. 6, have been analysed. The case studies have been taken from the first calibration of the PSM under mode III loading, in which 2D axis-symmetric models were adopted [16]. In the present analysis, the previous 2D geometries have been extruded about the tube axis to obtain the 3D geometries. It should be noted that only one quarter of each geometry has been analysed by taking advantage of the double anti-symmetry boundary conditions.

3D linear elastic analyses have been carried out by adopting ten-node, quadratic tetrahedral elements, in order to calculate the peak stress values. The mesh density ratio $a/d$ has been varied in a range from 1 to 10, $a$ being the tube thickness $t$ which ranged between 7 and 10 mm (see Table 1).

A nominal torsion shear stress equal to 1 MPa has been applied to each FE model at the tube side. After having solved the numerical model, the peak value of the (mode III) shear stress $\tau_{\theta z,\text{peak}}$ has been calculated at the vertex nodes belonging to the root or toe profile by adopting a local coordinate system $r$-$\theta$-$z$ rotated at each node according to Fig. 2. Then Eq. (3) has been applied to derive the average peak stress at each vertex node.

Finally, the exact values of the mode I NSIF $K_I$, mode II SIF $K_II$ and mode III NSIF $K_III$ to be input in Eq. (4a)-(4c), respectively, have been calculated by applying definitions (1a)-(1c) to the stress-distance numerical results obtained from 2D FE analyses by adopting very refined FE meshes (the size of the smallest element being on the order of $10^{-3}$ mm). Dealing with mode I and mode II problems, eight-node, quadratic quadrilateral elements (PLANE 183 of Ansys® element library) under plane strain conditions have been adopted, while concerning mode III problems, eight-node, quadratic quadrilateral harmonic elements (PLANE 83 of Ansys® element library) have been employed.

3 Results of FE analyses

The results obtained from the calibration of the PSM under mode I, mode II and mode III loadings by using ten-node tetra elements are shown in Figs. 7, 8 and 9, respectively. The figures report the parameters $K^*_FE$, $K^{**}_FE$ and $K^{***}_FE$, calculated from Eqs. (4a)-(4c), respectively, as functions of the mesh density ratio $a/d$. It should be noted that in each FE analysis, parameters $K^*_FE$, $K^{**}_FE$ and $K^{***}_FE$ exhibited a non-uniform distribution along the notch tip profile, given the variability of the average peak stress shown in Fig. 3. Accordingly, Figs. 7, 8 and 9 report the average value of the $K^*_FE$ parameters evaluated from each FE analysis along with the relevant bar, representing the range between minimum and maximum values calculated along the notch tip profile.

In the case of 3D problems under mode I loading, Fig. 7 shows that $K^*_FE \cong 1.01\pm15\%$ for $2\alpha$ equal to 0° or 90°, while $K^*_FE \cong 1.21\pm10\%$ when $2\alpha$ equals 135°. Convergence is obtained when $a/d \geq 3$ and 1 for $2\alpha$ equal to 0°, 90° and 135°, respectively.
Fig. 7. Calibration of $K_{FE}$ (Eq. 4a) for 10-node tetrahedral element under mode I loading: (a) $2\alpha = 0^\circ, 90^\circ$, (b) $2\alpha = 135^\circ$.

Dealing, then, with mode II loading, the obtained results are reported in Fig. 8, which shows that $K_{FE}^{**} \cong 1.63 \pm 20\%$. Convergence is obtained when the ratio $a/d \geq 1$.

Fig. 8. Calibration of $K_{FE}^{**}$ (Eq. 4b) for 10-node tetrahedral element under mode II loading.

Finally, concerning mode III loading, the obtained results are reported in Fig. 9, which shows that $K_{FE}^{***} \cong 1.37 \pm 10\%$ when root side ($2\alpha = 0^\circ$) is considered, while $K_{FE}^{***} \cong 1.75 \pm 5\%$ when toe side ($2\alpha = 135^\circ$) is of interest. Convergence is obtained when $a/d \geq 2$ both at weld root and toe sides.

Fig. 9. Calibration of $K_{FE}^{***}$ (Eq. 4c) for 10-node tetrahedral element under mode III loading: (a) $2\alpha = 0^\circ$, (b) $2\alpha = 135^\circ$.

A summary is reported in Table 2. It can be observed that the scatter bands of $K_{FE}$ parameters are wider at root side ($2\alpha = 0^\circ$) than at toe side ($2\alpha = 135^\circ$) for all loading modes. This is due to the presence of elements having significantly different shape and size (the FE size $d$ given as input being an average value) at the notch tip profile, especially in the case of cracks or weld root sides as compared to the case of open V-notches or weld toe sides. The non-uniform local mesh pattern has particular effect for cracks under mode II loading, the deviation of $K_{FE}^{**}$ being maximum and equal to $\pm 20\%$ (see Fig. 8).

Table 2. Summary of calibration of $K_{FE}^{*}$, $K_{FE}^{**}$ and $K_{FE}^{***}$ for 10-node tetrahedral elements (SOLID 187 of Ansys®).

<table>
<thead>
<tr>
<th>$2\alpha$ [°]</th>
<th>Mode I</th>
<th>Mode II</th>
<th>Mode III</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$K_{FE}^{*}$ $(a/d)_{\text{min}}$</td>
<td>$K_{FE}^{**}$ $(a/d)_{\text{min}}$</td>
<td>$K_{FE}^{***}$ $(a/d)_{\text{min}}$</td>
</tr>
<tr>
<td>0</td>
<td>1.01 ± 15%</td>
<td>1.63 ± 20%</td>
<td>1.37 ± 10%</td>
</tr>
<tr>
<td>90</td>
<td>1.01 ± 15%</td>
<td>/</td>
<td>/</td>
</tr>
<tr>
<td>135</td>
<td>1.21 ± 10%</td>
<td>/</td>
<td>1.75 ± 5%</td>
</tr>
</tbody>
</table>

4 Application to a case study

After presenting the calibration of the three-dimensional PSM based on tetra elements, an applicative example has
been considered (see Fig. 10). It is relevant to a large-scale steel welded structure, which represents a detail of a sluice gate having overall size on the order of tens of meters. The considered detail has size on the order of meters and is located at a geodetic height of about 10 meters referred to the free surface.

Fig. 10. Geometry (dimensions are in mm) and boundary conditions applied to the detail of the sluice gate. $\gamma$ is the water specific weight being equal to $10^4$ N/m$^3$, $h_A$ and $h_B$ are the geodetic height referred to the free surface and are equal to 9.232 and 10 meters, respectively.

The detail of Fig. 10 is characterised by many different welded geometries, including T- and cruciform-joints, both fillet and full-penetration welded. The plates thickness ranges between 10 and 58 mm.

The 3D PSM based on tetra elements, previously calibrated, has been applied to estimate the NSIFs at both weld toe and root sides of the considered large-scale steel welded structure. For the sake of brevity, only a selection of toe and root sides undergoing pure mode I loading have been analysed in the following. According to previous Fig. 7, the mesh density ratio $a/d$ must be greater than 1 to analyse weld toe sides and greater than 3 to analyse weld root sides. With the aim of comparing the solution time, two different rather coarse meshes (see Fig. 11) of ten-node tetra elements have been generated by adopting $a/d = 1$ and 3, respectively. The minimum plate thickness being equal to $2a = 10$ mm, the mesh density ratio $a/d = 1$ corresponds to $d = 5$ mm (see Fig. 11a), while $a/d = 3$ to a mesh size $d = 1.66$ mm (see Fig. 11b). The boundary conditions applied to both FE models are reported in Fig. 10. It should be noted that half of the hydrostatic pressure has been applied to the X-Z plane being active an anti-symmetry boundary condition. Both 3D FE models have been solved by taking advantage of Ansys HPC®, the global number of degrees of freedom (dof) being equal to 10 millions, for the case $a/d = 1$, while it reaches 140 millions, when $a/d = 3$. The performance of the computer cluster adopted to solve the FE analyses and the solution times are reported in Table 3.

Fig. 11. Coarse meshes of ten-node tetra elements generated in Ansys 18.2 environment and adopted to analyse (a) weld toe sides, being $a/d = 1$ according to Fig. 7b, and (b) weld root sides, being $a/d = 3$ according to Fig. 7a. Dof = degree of freedom.

Table 3. Performance of the computer cluster adopted to solve the FE analyses of Fig. 11 by using Ansys 18.2 HPC.

<table>
<thead>
<tr>
<th># d.o.f</th>
<th>CPU model</th>
<th># cores/ [Gb]</th>
<th>Solution elapsed time [s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>$10 \cdot 10^6$ (Fig. 11a)</td>
<td>Intel Xeon CPU E5-2665 2.40GHz</td>
<td>64/500</td>
<td>3660</td>
</tr>
<tr>
<td>$140 \cdot 10^6$ (Fig. 11b)</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figures 12 and 13 report the distribution of the mode I NSIF estimated at selected toe and root sides according to the 3D PSM previously calibrated. More in detail, the NSIF $K_1$ has been estimated at the selected toe side (Fig. 12) by applying Eq. (2a) with $K_{FE}^{*} = 1.21$ to the average peak stress values calculated from the FE model of Fig. 11a. On the other hand, the SIF $K_1$ has been estimated at the selected root side (Fig. 13) by applying Eq. (2a) with $K_{FE}^{*} = 1.01$ to the average peak stress values calculated from the FE model of Fig. 11b.
For comparison purposes, the mode I NSIF values estimated at the selected toe and root sides according to a shell-to-solid technique [21] have been included in Figs. 12 and 13. A quite good agreement can be observed, the maximum deviation being equal to approximately 10% at the toe side and approximately 15% at the root side.

5 Conclusions

The peak stress method (PSM) employs the singular, linear elastic peak stresses evaluated at the notch tip by means of FE analyses with coarse meshes to rapidly estimate the mode I, mode II and mode III NSIFs. Three calibration constants are needed, namely $K_{FE}^*$ (Eq. (4a)), $K_{FE}^{-}$ (Eq. (4b)) and $K_{FE}^{-}$ (Eq. (4c)), respectively.

Originally, the PSM was calibrated by using 2D plane or 3D brick elements, taking advantage of the submodeling technique. In the present contribution the PSM has been calibrated under mode I, II and III loadings by using ten-node tetra elements, which are able to directly discretize complex 3D geometries without the need for submodels. The following conclusions can be drawn:

- It has been shown that under mode I loading the constant $K_{FE}^*$ to use in Eq. (2a) is 1.01±15% for $2\alpha = 0^\circ$, 90°, while it equals 1.21±10% when $2\alpha = 135^\circ$. Convergence is obtained when the mesh density ratio $a/d \geq 3$ for $2\alpha$ equal to $0^\circ$, 90° and $a/d \geq 1$ for $2\alpha = 135^\circ$.

- Dealing with mode II loading, it has been shown that the constant $K_{FE}^{-}$ to use in Eq. (2b) is 1.63±20% for a mesh density ratio $a/d \geq 1$.

- Concerning mode III loading, $K_{FE}^{-}$ to use in Eq. (2c) results 1.37±10% at root side ($2\alpha = 0^\circ$) and 1.75±5% at the toe side ($2\alpha = 135^\circ$). Convergence occurs when $a/d \geq 2$ both at root and toe sides.

- Finally, an applicative example has been considered, which is relevant to a large-scale steel welded structure, having overall size on the order of several meters. The Ansys High Performance Computing (HPC) has been adopted to solve two 3D FE models, by employing ten-node tetra elements with global element size $d = 5$ and 1.67 mm, respectively. The solution time was on the order of 200 s for the FE model with $d = 5$ mm, i.e. 10 millions degrees of freedom, while about 3500 s were needed to solve the FE model with $d = 1.67$ mm, i.e. 140 millions degrees of freedom. The mode I NSIF values estimated at the toe and root sides according to the PSM have been compared with those calculated by adopting a shell-to-solid technique, showing a quite good agreement.

- Because of the relatively coarse FE analyses required and simplicity of post-processing the calculated peak stresses, the PSM based on three-dimensional models of ten-node tetra elements might be useful in the everyday design practice, even when large-scale structures are considered.

EnginSoft SpA (Padova, Italy) is gratefully acknowledged for making Ansys HPC available for this project.

References