



1st Virtual European Conference on Fracture

Automated fatigue strength assessment of arc-welded structures according to the Peak Stress Method

Giovanni Meneghetti*, Alberto Campagnolo, Alberto Visentin

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

Abstract

The Peak Stress Method (PSM) is an approximate, FE-oriented application of the notch stress intensity factor (NSIF) approach to fatigue design of welded joints. This approach is based on the singular linear elastic peak stresses calculated from coarse FE models. The PSM was calibrated by using 2D, four-node plane quadrilateral elements; 3D, eight-node brick elements and 3D, four-node or ten-node tetra elements. By adopting the averaged Strain Energy Density (SED) as a fatigue strength criterion, a design stress, the so-called equivalent peak stress, has been defined and adopted in conjunction with a reference design fatigue curve to estimate the fatigue life of welded structures. The equivalent peak stress has successfully been adopted to assess the fatigue strength of arc-welded joints made of either structural steels or aluminum alloys and subjected to axial, torsion or multiaxial fatigue loadings. In the present work, a subroutine, named ANSYS-PSM, has been developed in the post-processing environment of ANSYS® FE code to automate the PSM for 2D or 3D FE models. After having briefly recalled the theoretical background of the PSM and the procedure to apply it step by step, the paper presents the adopted programming languages and the workflow of the developed subroutine.

© 2020 The Authors. Published by Elsevier B.V.

This is an open access article under the CC BY-NC-ND license (<https://creativecommons.org/licenses/by-nc-nd/4.0>)

Peer-review under responsibility of the European Structural Integrity Society (ESIS) ExCo

Keywords: Welded Joints; Fatigue; Peak Stress Method; Coarse mesh; Subroutine

* Corresponding author. Tel.: +39 049 8276751; fax: +39 049 8276785.

E-mail address: giovanni.meneghetti@unipd.it

1. Introduction

According to the local fatigue approach based on the NSIF-parameters (Lazzarin et al., 2004; Lazzarin and Tovo, 1998; Radaj et al., 2006), a worst-case condition corresponding to a sharp V-notch having null tip radius ($\rho = 0$) is assumed both at the weld toe and at the weld root of a joint detail, the toe and root sides being modelled with opening angles of 135° and 0° , respectively, as highlighted in Fig. 1a. Given these idealisations and assuming a linear-elastic behaviour of the material, an external load applied to a welded structure generates singular stress fields in the local material regions closed to both toe and root sides. The NSIF-parameters, which can be defined according (Gross and Mendelson, 1972) by means of Eq. (1), allow to quantify the intensity of these stress distributions.

$$K_i = \sqrt{2\pi} \cdot \lim_{r \rightarrow 0} \left[(\sigma_{jk})_{\theta=0} \cdot r^{1-\lambda_i} \right] \quad \text{where } i=1,2,3 \quad \text{and } \sigma_{jk} = \sigma_{\theta\theta}, \tau_{r\theta}, \tau_{\theta z} \text{ respectively} \quad (1)$$

The linear elastic, local stress components $\sigma_{\theta\theta}$, $\tau_{r\theta}$ and $\tau_{\theta z}$ are calculated close to the notch tip ($r \rightarrow 0$) and along the notch bisector line ($\theta=0$), as shown in the example of Fig. 1b. Finally, parameters λ_1 , λ_2 and λ_3 are correlated to the stress singularity degrees (Qian and Hasebe, 1997; Williams, 1952) and are functions of the opening angle 2α of the analysed V-notch (see Table 1).

Table 1: Values of parameters depending on the notch opening angle 2α .

| 2α ($^\circ$) | $\lambda_1^{(a)}$ | $\lambda_2^{(a)}$ | $\lambda_3^{(a)}$ | Aluminium $\nu = 0.33^{(b)}$ | | | Steel $\nu = 0.3^{(b)}$ | | |
|---------------------------|-------------------|-------------------|-------------------|------------------------------|-------|-------|-------------------------|-------|-------|
| | | | | e_1 | e_2 | e_3 | e_1 | e_2 | e_3 |
| 0 | 0.500 | 0.500 | 0.500 | 0.125 | 0.337 | 0.423 | 0.134 | 0.341 | 0.414 |
| 90 | 0.545 | 0.909 | 0.667 | 0.138 | 0.168 | 0.318 | 0.146 | 0.168 | 0.310 |
| 120 | 0.616 | - | 0.750 | 0.124 | - | 0.282 | 0.130 | - | 0.276 |
| 135 | 0.674 | - | 0.800 | 0.113 | - | 0.265 | 0.117 | - | 0.259 |

^(a): values from (Lazzarin et al., 2008)

^(b): values calculated under plain strain conditions

Lazzarin and collaborators (Lazzarin et al., 2008; Lazzarin and Zambardi, 2001; Livieri and Lazzarin, 2005) assumed the strain energy density (SED) averaged over a structural volume surrounding the weld root or the weld toe as a fatigue strength criterion. They assumed a structural volume having circular shape with radius R_0 (see Fig. 1c) and provided the closed-form expression of the averaged SED parameter as a function of the relevant NSIFs. Dealing with a general multiaxial fatigue loading condition (mixed mode I+II+III loading, see Fig. 1), the SED averaged over the control volume can be expressed as follows (Lazzarin et al., 2008):

$$\Delta \bar{W} = \left\{ c_{w1} \frac{e_1}{E} \left[\frac{\Delta K_1}{R_0^{1-\lambda_1}} \right]^2 + c_{w2} \frac{e_2}{E} \left[\frac{\Delta K_2}{R_0^{1-\lambda_2}} \right]^2 + c_{w3} \frac{e_3}{E} \left[\frac{\Delta K_3}{R_0^{1-\lambda_3}} \right]^2 \right\} \quad (2)$$

In Eq. (2), E represents the material modulus of elasticity; e_1 , e_2 and e_3 are parameters dependent on the sharp notch geometry and on the material, through the opening angle 2α and the Poisson's ratio ν (see Table 1), respectively, while ΔK_1 , ΔK_2 and ΔK_3 are the range values of the NSIFs relevant to mode I, II and III, respectively. R_0 represents the structural volume size and results equal to 0.12 mm and 0.28 mm for arc-welded joints made of aluminium alloys and structural steels (Lazzarin et al., 2003; Livieri and Lazzarin, 2005), respectively. Finally, parameters c_{wi} account for the mean stress effect and depend on the load ratio R_i of the i -th mode of loading ($i = 1, 2$ or 3) according to the following expression (Lazzarin et al., 2004):

$$c_{wi}(R_i) = \begin{cases} \frac{1+R_i^2}{(1-R_i)^2} & \text{if stress-relieved and } -1 \leq R_i \leq 0 \\ \frac{1-R_i^2}{(1-R_i)^2} & \text{if stress-relieved and } 0 \leq R_i < 1 \\ 1 & \text{if as-welded for any } R \text{ value} \end{cases} \quad \text{where } i = 1, 2 \text{ or } 3 \quad (3)$$

In particular, the parameter c_w equals 0.5 and 1 for fully reversed ($R = -1$) and pulsating ($R = 0$) loadings, respectively. It should be noted that welded joints loaded in the as-welded conditions are almost not sensitive to mean stresses, according to design standards (*Eurocode 3: Design of steel structures – part 1–9: Fatigue, 2005*), therefore Eq. (2) with $c_{wi}=1$ should be applied.

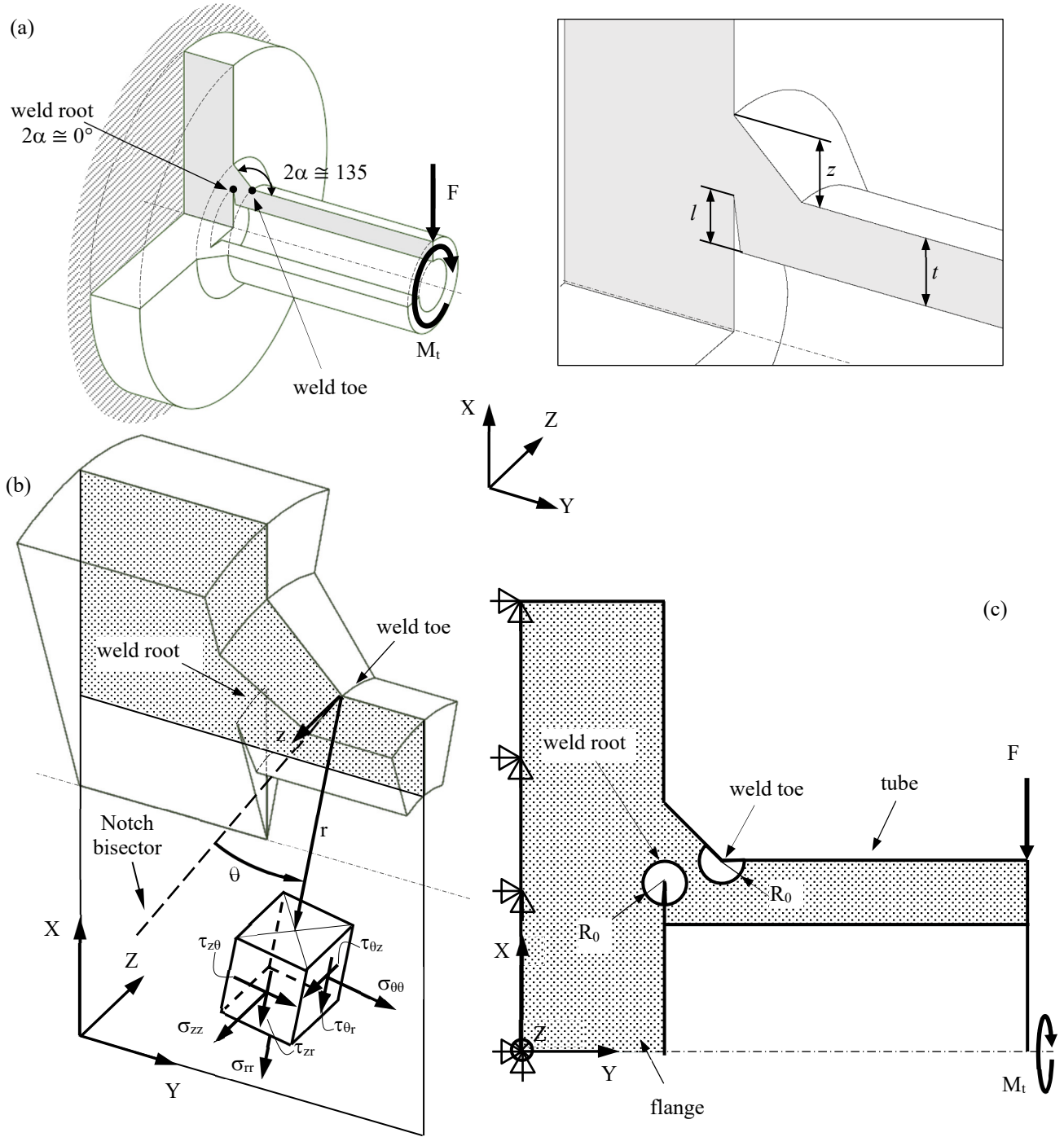


Fig. 1. (a) Assumptions of the NSIF-based approach in fatigue design of welded joints referring to a partial-penetration tube-to-flange welded joint under combined bending and torsion fatigue loading. (b) Cylindrical reference system (r, θ, z) centred at the weld toe and local stress components. (c) The structural volume of radius R_0 centred either at the weld toe or at the weld root according to the averaged SED approach.

2. Peak Stress Method (PSM)

The Peak Stress Method (PSM) is a rapid, numerical tool to rapidly estimate the NSIF-terms K_1 , K_2 and K_3 , taking advantage of the opening, in-plane shear and out-of-plane shear peak stresses, respectively, calculated from a linear elastic FE analysis with coarse mesh, as shown in the example of Fig. 2 for a tube-to-flange joint. The estimated NSIF values can be obtained from the following expressions (Meneghetti, 2013, 2012; Meneghetti and Lazzarin, 2007):

$$K_1 \cong K_{FE}^* \cdot \sigma_{\theta\theta,\theta=0,\text{peak}} \cdot d^{1-\lambda_1}; \quad K_2 \cong K_{FE}^{**} \cdot \tau_{r\theta,\theta=0,\text{peak}} \cdot d^{1-\lambda_2}; \quad K_3 \cong K_{FE}^{***} \cdot \tau_{\theta z,\theta=0,\text{peak}} \cdot d^{1-\lambda_3} \quad (4)$$

In previous expressions, $\sigma_{\theta\theta,\theta=0,\text{peak}}$, $\tau_{r\theta,\theta=0,\text{peak}}$ and $\tau_{\theta z,\theta=0,\text{peak}}$ are peak stresses defined in a local cylindrical coordinate system centred at the node at the V-notch tip; z -direction is tangential to the notch tip line; the θ -direction originates from the notch bisector line and r is the radial direction. The subscript ' $\theta=0$ ' highlights the direction along which local stresses must be evaluated; as an example, $\sigma_{\theta\theta,\theta=0,\text{peak}}$ means that the opening stress acts in normal direction with respect to the notch bisector, as shown in Fig. 2. Parameter d in Eq. (4) represents the average size of the finite elements which the FE analyst gives as input to the free mesh generation algorithm of the employed numerical software. Finally, parameters K_{FE}^* , K_{FE}^{**} and K_{FE}^{***} are dependent on the (Meneghetti et al., 2018): (i) element type and formulation; (ii) FE mesh pattern and (iii) procedure employed by the numerical software to extrapolate stresses at nodes.

The estimation of NSIF-terms by using the PSM according to Eq. (4) is advantageous, since: (i) coarse FE meshes can be employed; moreover, (ii) only the linear-elastic peak stresses calculated at the notch tip are necessary, instead of a complete set of stress-distance numerical results as required to apply Eq. (1).

A state-of-the-art review of the PSM has recently been published in (Meneghetti and Campagnolo, 2020), which the reader is referred to for additional details about the method.

2.1. Calibrated FE types

The calibration of parameters K_{FE}^* , K_{FE}^{**} and K_{FE}^{***} has been performed in previous literature by using several 2D and 3D FE types (Campagnolo et al., 2019b; Campagnolo and Meneghetti, 2018; Meneghetti, 2013, 2012; Meneghetti and Guzzella, 2014; Meneghetti and Lazzarin, 2007; Visentin, 2020). Table 2 summarises the values of PSM coefficients K_{FE}^* , K_{FE}^{**} and K_{FE}^{***} as calibrated by using 4-node plane, 8-node brick and 4-node or 10-node tetra elements of Ansys® element library.

In order to apply the PSM by adopting tetra elements, the variation of the peak stress along the notch tip profile due to intrinsically irregular free mesh pattern (see (Meneghetti and Campagnolo, 2020)) has been reduced by defining an average peak stress according to Eq. (5). More in detail, the average peak stress at the generic node $n=k$ is the moving average of peak stresses calculated on three adjacent vertex nodes, i.e. $n=k-1$, k and $k+1$, according to the following expression (Campagnolo and Meneghetti, 2018):

$$\bar{\sigma}_{ij,\text{peak},n=k} = \frac{\sigma_{ij,\text{peak},n=k-1} + \sigma_{ij,\text{peak},n=k} + \sigma_{ij,\text{peak},n=k+1}}{3} \Bigg|_{n=\text{node}} \quad (5)$$

Moreover, the calibration of PSM based on tetra elements (Campagnolo et al., 2019b; Campagnolo and Meneghetti, 2018) requires that peak stresses evaluated at the nodes of the notch tip profile, which lay on the free surface of the analysed structure, are neglected, since their value can be affected by distorted mesh patterns (see Fig. 2). Furthermore, only peak stresses calculated at vertex nodes of tetra elements must be used in Eq. (5) to compute the average peak stress; therefore, when adopting 10-node tetra elements, peak stresses referred to mid-side nodes located along the notch tip profile must be neglected (see Fig. 2). Accordingly, PSM-parameters K_{FE}^* , K_{FE}^{**} and K_{FE}^{***} have been calibrated using either 4-node (Campagnolo et al., 2019b) or 10-node (Campagnolo and Meneghetti, 2018) tetra elements and by substituting the average peak stresses according to Eq. (5), i.e. $\bar{\sigma}_{\theta\theta,\theta=0,\text{peak}}$, $\bar{\tau}_{r\theta,\theta=0,\text{peak}}$ and $\bar{\tau}_{\theta z,\theta=0,\text{peak}}$ withing Eq. (4), in place of peak stresses $\sigma_{\theta\theta,\theta=0,\text{peak}}$, $\tau_{r\theta,\theta=0,\text{peak}}$ and $\tau_{\theta z,\theta=0,\text{peak}}$, respectively.

2.2. Range of PSM applicability

The calibration of PSM parameters K_{FE}^* , K_{FE}^{**} and K_{FE}^{***} has been performed under the conditions discussed in

(Meneghetti and Campagnolo, 2020) and briefly recalled in the following (see also Table 2):

- a range of notch opening angles has been considered:
 - for mode I and mode III loadings: 2α has been varied in the range $0^\circ - 135^\circ$ being the typical opening angles at the weld root and toe sides, respectively. It is worth noting that in previous contributions the same calibration constants have been adopted to analyse weld toe sides having opening angles up to about 150° (Meneghetti et al., 2015);
 - for mode II loading: only $2\alpha = 0^\circ$ (Campagnolo et al., 2019b; Campagnolo and Meneghetti, 2018; Meneghetti, 2012) and 90° (Visentin, 2020) have been investigated, being the typical cases of the weld root side without or with a gap, respectively; while the case $2\alpha = 135^\circ$ has not been analysed since mode II stresses are not singular in this case.
- the FE mesh pattern in the proximity of the notch or crack tip must satisfy the following conditions:
 - 4-node plane or 8-node brick elements: the number of elements sharing the node located at the singularity point must be 4 when $2\alpha \leq 90^\circ$ (e.g. at the weld root $2\alpha \cong 0^\circ$) and 2 when $2\alpha > 90^\circ$ (e.g. at the toe side $2\alpha \cong 135^\circ$).
 - 4-node or 10-node tetra elements: it is not required to check the number of elements sharing the node located at the notch tip.
- a minimum mesh density ratio a/d must be adopted to define the free mesh pattern according to PSM, a being the characteristic size of the considered notch (see definition in Table 2 and Fig. 2, (Meneghetti and Campagnolo, 2020)). The minimum value of a/d depends on the adopted FE type, loading mode and the opening angle of the analysed notch, as highlighted in Table 2.

2.3. Evaluation of peak stresses by FE analyses according to PSM

Several strategies are available to apply the PSM and to calculate the peak stresses by post-processing 2D or 3D FE models, as it has been discussed in detail in (Meneghetti and Campagnolo, 2020). For sake of brevity, only two fundamental remarks are recalled here:

- peak stresses $\sigma_{\theta\theta,\theta=0,\text{peak}}$, $\tau_{r\theta,\theta=0,\text{peak}}$ and $\tau_{\theta z,\theta=0,\text{peak}}$ in Eq. (4) are defined in a local cylindrical coordinate system having centre at a FE node located along the V-notch tip profile, z -direction tangent to the notch tip profile, θ -direction originating from the notch bisector line and r being the radial direction. When a 2D or 3D PSM is adopted for rectilinear weld toes and weld root lines, the definition of a single cylindrical coordinate system is sufficient for calculating peak stresses at all nodes of the weld toe and weld root. However, in the most general case of 3D joint geometries with curvilinear weld toes and weld root lines, a dedicated local cylindrical coordinate system must be defined at each node because the z -direction must change from node to node to ensure tangency to the notch tip line.
- to apply the 3D PSM with the Ansys® FE code, the ‘FULL graphics’ option must be activated to evaluate the peak stress in the post-processing environment.

2.4. Equivalent stress for fatigue design

It has been shown in Eq. (2) that the averaged SED can be expressed as a function of NSIF-terms K_1 , K_2 and K_3 , which the PSM allows to rapidly estimate through Eq. (4). Accordingly, the averaged SED can be rewritten as a function of the relevant peak stresses; moreover, by introducing the SED value for an equivalent uniaxial plane strain state, i.e. $W = (1 - \nu^2) \sigma_{\text{eq,peak}}^2 / 2E$, an equivalent peak stress can be defined by Eq. (6a) and (6b) (Campagnolo et al., 2019a; Meneghetti et al., 2017a, 2017b):

$$\Delta\sigma_{\text{eq,peak}} = \sqrt{c_{w1} \cdot f_{w1}^2 \cdot \Delta\sigma_{\theta\theta,\theta=0,\text{peak}}^2 + c_{w2} \cdot f_{w2}^2 \cdot \Delta\tau_{r\theta,\theta=0,\text{peak}}^2 + c_{w3} \cdot f_{w3}^2 \cdot \Delta\tau_{\theta z,\theta=0,\text{peak}}^2} \quad \text{PSM Plane-4 or Brick-8} \quad (6a)$$

$$\Delta\sigma_{\text{eq,peak}} = \sqrt{c_{w1} \cdot f_{w1}^2 \cdot \Delta\bar{\sigma}_{\theta\theta,\theta=0,\text{peak}}^2 + c_{w2} \cdot f_{w2}^2 \cdot \Delta\bar{\tau}_{r\theta,\theta=0,\text{peak}}^2 + c_{w3} \cdot f_{w3}^2 \cdot \Delta\bar{\tau}_{\theta z,\theta=0,\text{peak}}^2} \quad \text{PSM Tetra-4 or Tetra-10} \quad (6b)$$

Previous equations introduce parameters f_{wi} (where $i = 1, 2, 3$ represents the loading mode) which account for peak stress averaging inside the material-structural volume having size R_0 (see Fig. 1c), and they are defined as follows (Meneghetti et al., 2017a, 2017b):

$$f_{wi} = K_{FE} \cdot \sqrt{\frac{2e_i}{1-\nu^2}} \cdot \left(\frac{d}{R_0}\right)^{1-\lambda_i} \quad \text{where } i = 1, 2, 3 \quad (7)$$

Equations (4) and (7) show that both peak stresses and coefficients f_{wi} are functions of the average element size d employed in the FE model to apply the PSM; however, the equivalent peak stress defined in Eqs. (6a) and (6b), which includes the peak stresses multiplied by the relevant f_{wi} s, results to be independent of the average element size d .

Table 2. Summary of parameters K_{FE}^* , K_{FE}^{**} and K_{FE}^{***} and mesh density requirements a/d to apply PSM with finite elements of Ansys® element library.

| Loading | FE type [#] | PSM parameters | $2\alpha = 0^\circ$ | $2\alpha = 90^\circ$ | $2\alpha = 120^\circ$ | $2\alpha = 135^\circ$ | $a - \text{root side}^\circ$ | $a - \text{toe side}^\circ$ |
|----------|----------------------|--|---|---|-----------------------|-----------------------|------------------------------|-----------------------------|
| Mode I | Plane-4 Brick-8 | K_{FE}^* (a/d) _{min} FE at notch tip [^] | 1.38±3% 3 4 | | 4 | 2 | 2 | min { l, z } t |
| | Tetra-4 | K_{FE}^* (a/d) _{min} FE at notch tip [^] | 1.75±22% 3 <i>not to be checked</i> | | | | | |
| | Tetra-10 | K_{FE}^* (a/d) _{min} FE at notch tip [^] | 1.05±15% 3 <i>not to be checked</i> | | | 1.21±10% 1 | | |
| Mode II | Plane-4 Brick-8 | K_{FE}^{**} (a/d) _{min} FE at notch tip [^] | 3.38±3% 14 4 | 2.62±10% 10 4 | - - - | - - - | | min { l, z } - |
| | Tetra-4 | K_{FE}^{**} (a/d) _{min} FE at notch tip [^] | 2.65±15% 3 <i>not to be checked</i> | 2.90±10% 1 <i>not to be checked</i> | - - - | - - - | | |
| | Tetra-10 | K_{FE}^{**} (a/d) _{min} FE at notch tip [^] | 1.63±20% 1 <i>not to be checked</i> | 2.65±10% 1 <i>not to be checked</i> | - - - | - - - | | |
| Mode III | Plane-4 Brick-8 | K_{FE}^{***} (a/d) _{min} FE at notch tip [^] | 1.93±3% 12 4 | - - - | - - - | | 3 2 | min { l, z } t |
| | Tetra-4 | K_{FE}^{***} (a/d) _{min} FE at notch tip [^] | 2.50±15% 5 <i>not to be checked</i> | | | | | |
| | Tetra-10 | K_{FE}^{***} (a/d) _{min} FE at notch tip [^] | 1.37±15% 3 <i>not to be checked</i> | | | 1.70±10% 3 | | |

- # FE of Ansys® code: Plane-4 = PLANE 182 (K-option 1 set to 3) or PLANE 25,
Brick 8 = SOLID 185 (K-option 2 set to 3), Tetra 4 = SOLID 285, Tetra 10 = SOLID 187
^ number of finite elements which share the node at the notch tip
° l, z, t are defined in Fig. 1a

2.5. Criterion to select the reference fatigue design curve

According to (Campagnolo et al., 2019a; Meneghetti et al., 2019), the proper master curve for the fatigue design of arc-welded joints should be selected based on the relative contributions from the local shear and local normal stresses. To be consistent within the adopted fatigue local approach, a local biaxiality ratio λ has been defined in (Campagnolo et al., 2019a; Meneghetti et al., 2019) as the ratio between the SED contributions due to mode II/III shear and mode I. Then, λ can be expressed as a function of the peak stresses according to Eq. (8a) and (8b):

$$\lambda = \frac{c_{w2} \cdot f_{w2}^2 \cdot \Delta\tau_{r\theta, \theta=0, peak}^2 + c_{w3} \cdot f_{w3}^2 \cdot \Delta\tau_{\theta z, \theta=0, peak}^2}{c_{w1} \cdot f_{w1}^2 \cdot \Delta\sigma_{\theta\theta, \theta=0, peak}^2} \quad \text{PSM Plane-4 or Brick-8} \quad (8a)$$

$$\lambda = \frac{c_{w2} \cdot f_{w2}^2 \cdot \Delta\bar{\tau}_{r\theta, \theta=0, peak}^2 + c_{w3} \cdot f_{w3}^2 \cdot \Delta\bar{\tau}_{\theta z, \theta=0, peak}^2}{c_{w1} \cdot f_{w1}^2 \cdot \Delta\bar{\sigma}_{\theta\theta, \theta=0, peak}^2} \quad \text{PSM Tetra-10 or Tetra-4} \quad (8b)$$

It is worth noting that $\lambda = 0$ corresponds to a local pure mode I loading condition, while $\lambda \rightarrow \infty$ when only local mode II/III shear stresses are present; finally values between 0 and ∞ correspond to mixed mode opening-shear loading conditions. The criterion to select the reference fatigue design curve as a function of the joint material and local loading condition has been defined in (Meneghetti and Campagnolo, 2020) and it is reported in Table 3 for arc-welded joints made of aluminium alloys and structural steels.

Table 3: Criterion for selecting the reference PSM-based fatigue design curve for arc-welded joints

| Class of materials | Thickness, T [mm] | λ Eq. (8) | $\Delta\sigma_{eq, peak, A, 50\%}$ [MPa] | k | T_σ |
|--------------------|------------------------------|-------------------|--|-----|------------|
| Aluminium alloys | $T \geq 5$ mm | $\lambda = 0$ | 123 | 3.8 | 1.80 |
| | $T \geq 5$ mm | $\lambda > 0$ | 123 | 6.5 | 1.80 |
| | $3 \text{ mm} \leq T < 5$ mm | $\lambda \geq 0$ | | | |
| Structural steels | $T \geq 2$ mm | $\lambda = 0$ | 214 | 3 | 1.90 |
| | $T \geq 2$ mm | $\lambda > 0$ | 354 | 5 | 1.90 |

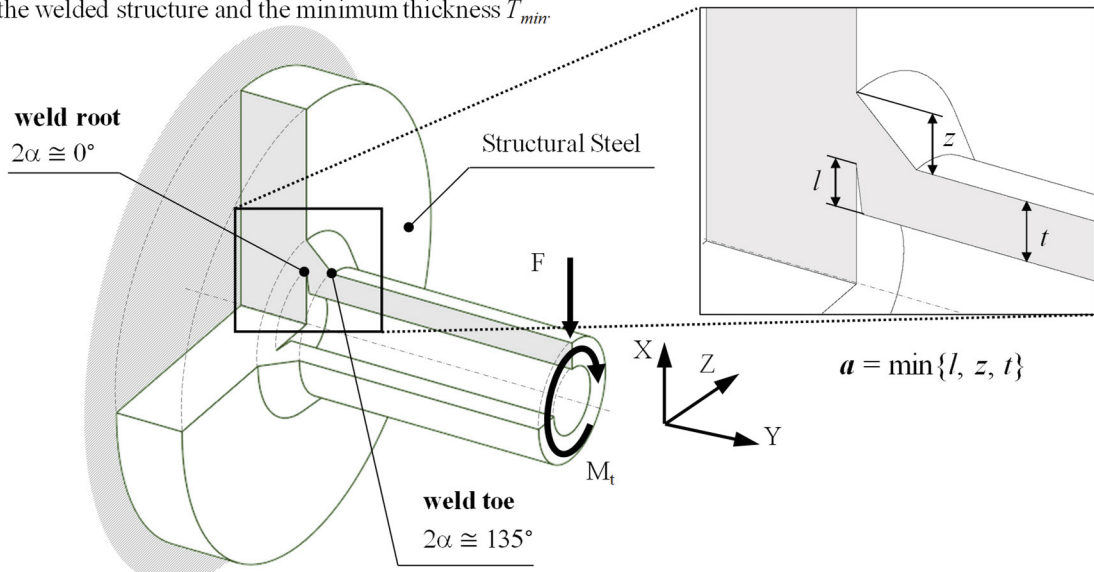
2.6. Fatigue strength assessment of arc-welded structures according to the PSM

All steps to manually apply the PSM for the fatigue strength assessment of arc-welded structures have been briefly recalled in previous sections and are schematically reported also in Fig. 2. The reader is referred to (Meneghetti and Campagnolo, 2020) for a comprehensive review on the application of the PSM.

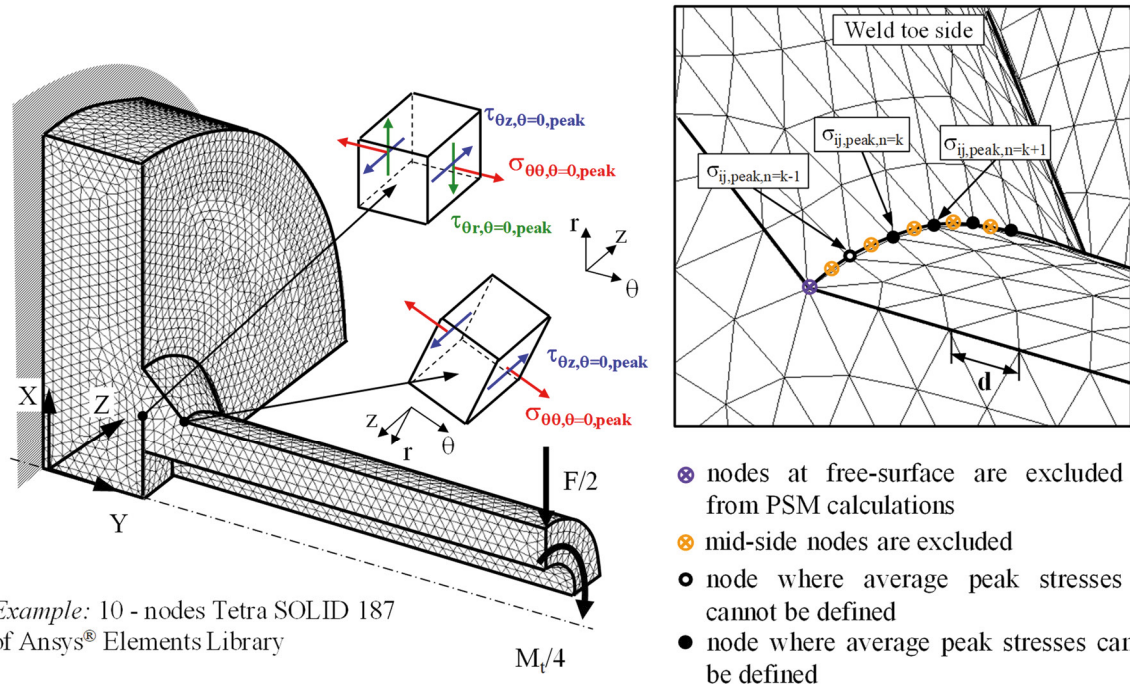
In the following, a subroutine, named ANSYS-PSM, developed in the post-processing environment of ANSYS® FE code to automate the application of the PSM to 2D or 3D FE models will be presented.

Step-by-step procedure to manually apply the Peak Stress Method (PSM)

- 1) Define the notch opening angle 2α at either weld toe or root sides. Moreover, define the characteristic size a of the welded structure and the minimum thickness T_{min}



2) Adopt a global element size d to generate a free mesh and fulfil the minimum mesh density ratio (a/d) as a function of the adopted element type, notch opening angle and loading mode, as reported in Table 2.



For each node of the considered location (weld toe or root side), repeat the following steps:

- 3) Define a properly-oriented local coordinate system r - θ - z centered at the considered node and calculate peak stress components referring to the local coordinate system
- 4) When adopting FE meshes of tetra elements, calculate the moving average peak stress components by means of Eq. (5)
- 5) Evaluate the stress singularity exponents λ_i and the averaged SED coefficients e_i as a function of the notch opening angle of the considered location (weld toe or root side) and of the material (see Table 1)
- 6) Evaluate the PSM parameters f_{wi} from Eq. (7) and the mean stress correction factors c_{wi} from Eq. (3), for each loading mode.
- 7) Evaluate the equivalent peak stress from Eq. (6) and the local biaxiality ratio λ from Eq. (8)
- 8) Select the proper PSM fatigue design curve according to the criterion defined in Table 3
- 9) By comparing the equivalent peak stress calculated at point (7) with the selected PSM fatigue design curve, estimate the fatigue life with a desired survival probability.

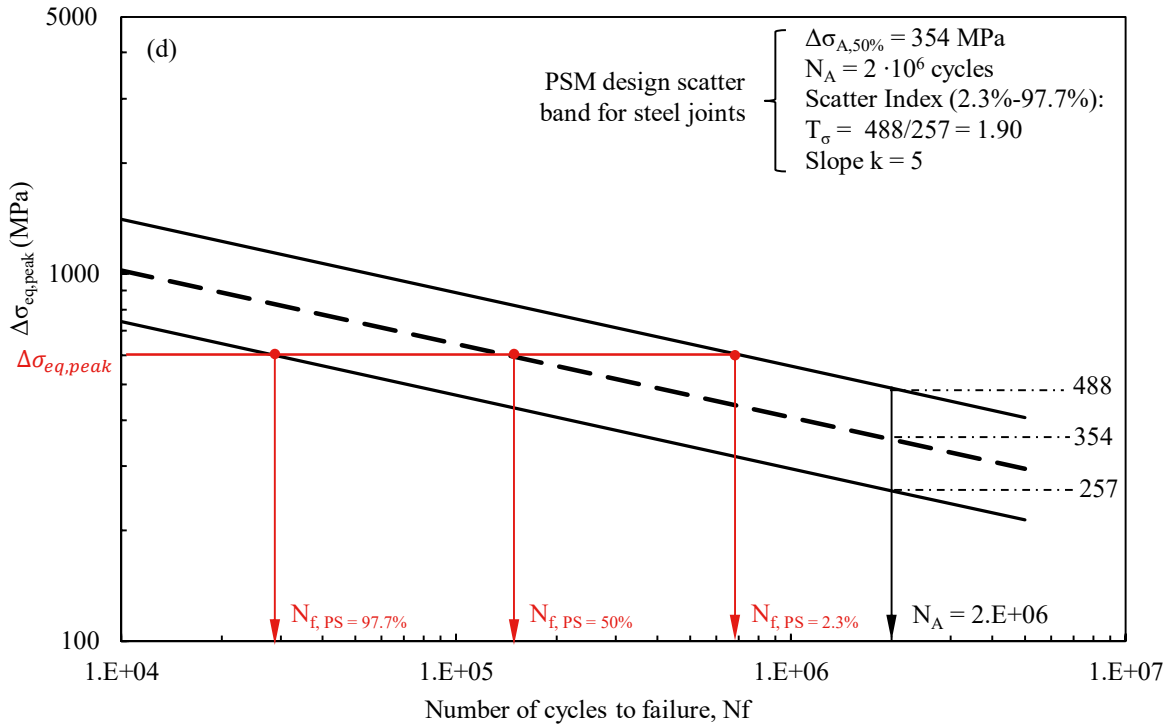


Fig. 2 Step-by-step procedure to manually apply the PSM for the fatigue strength assessment of a welded structure.

3. Programming languages available in ANSYS® Mechanical APDL

Automation grounds its roots on data computing and elaboration, achieved through programming. An automated routine can be described as a combination of complementary computer – driven actions, each one defining a different step of a complex procedure, in order to satisfy successive requirements, complete specific operations or perform recursive tasks. These actions are typically defined taking advantage of the most appropriate coding language with respect to the application context of interest.

The proposed routines find place in the ANSYS® Mechanical APDL environment, which offers the possibility to access scripting capabilities through the main supported languages of the native software itself, namely ANSYS® Parametric Design Language (APDL) and ANSYS® User Interface Design Language (UIDL). Indeed, each feature and function of ANSYS® environment can be programmed and handled through specific commands.

3.1. ANSYS® Parametric Design Language (APDL)

ANSYS® Parametric Design Language, commonly known as APDL, is the parametric coding language natively featured in ANSYS® environment. Each one of the functions manually accessible through ANSYS® interface is associated to one or more APDL commands, building a bridge between the user and the FE code of the software itself. In order to define custom tasks and automated workflows in ANSYS®, APDL command lines can be combined into an APDL Macro, namely a script defining a series of operations which the software will execute subsequently.

More than seven thousand APDL command lines, included in 15 APDL Macros, define the back-end core structure of ANSYS – PSM program described in this work, allowing either automation of the different tasks defining the Peak Stress Method operational workflow and creation of custom procedures for results post-processing and visualization.

3.2. ANSYS® User Interface Design Language (UIDL)

ANSYS® User Interface Design Language, also known as UIDL, is a high-level coding language natively supported in ANSYS® visual environment, aimed at designing and developing programmable graphical interfaces and features inside the native GUI (Graphical User Interface) of the software itself. UIDL is a programming language that enables to define and add new components to the ANSYS® Graphical User Interface, or customize existing ones. Among customizable components are: features on the ANSYS® Main Menu tree, quick interactive buttons and ribbons, dialog prompts and entity picking functions, custom help sections and call-backs. Parametric Design Language (APDL) and User Interface Design Language (UIDL) can be interfaced, mixed and used together in order to create a bridge between the user and the different functions in ANSYS® by means of the GUI. In order to properly define a custom interface, UIDL command lines need to be combined into a Control File.

More than six hundred UIDL command lines, included in 2 UIDL Control Files, define the front – end structure of ANSYS - PSM program presented in this work, allowing extension of ANSYS® standard Main Menu to include custom dedicated interactive graphical interfaces.

3.3. Tool Command Language and Toolkit (Tcl/Tk)

Tcl/Tk (Tool Command Language and ToolKit) is a dynamic high-level programming language, suitable for a wide range of implementations, most involving front-end features creation and web/desktop applications development, networking and testing. Tcl/Tk is also a cross-platform language, compatible with different environments and extensible. The main interest in the development of Tcl/Tk scripts within ANSYS® is due to the possibility of directly implementing Tcl code and Tk features within ANSYS® GUI and FE code, taking advantage of the native support offered by the software through APDL/UIDL integration. All features developed in Tcl/Tk language can be therefore imported within ANSYS® environment, allowing definition of custom widgets and contents to display information or data. Moreover, all inputs generated within an imported Tcl/Tk script can be easily converted to parameters or used to control specific ANSYS® FE functions.

4. Fitting equations to automate the calculation of stress singularities and SED parameters

In order for the Peak Stress Method to be implemented in a fully-automated computational way, the parameters depending on the material and local geometry of the analysed welded structure, namely the stress singularity exponents ($1-\lambda_i$) and the coefficients (e_i) for the averaged SED calculation, should be calculable in a rapid and easy way. Indeed, both parameters are necessary to calculate the coefficients f_{wi} by means of Eq. (7). Accordingly, the aim of the numerical elaborations described in the following is to provide a fitting equation for each parameter to be implemented in the ANSYS – PSM routines, allowing the PSM to be applied to welded structures in a wide range of geometries and materials.

4.1. Stress singularity exponents ($1-\lambda_i$) – fitting equations

The singularity exponents of mode I, II and III local stresses are functions of λ_i parameters, which have been defined by Williams (Williams, 1952) and Qian and Hasebe (Qian and Hasebe, 1997) dealing with mode I+II and mode III loadings, respectively, as the first positive solution ($\lambda_i > 0$) of the following transcendent equations:

$$\sin(2\gamma\lambda_1) + \lambda_1 \sin(2\gamma) = 0 \quad \sin(2\gamma\lambda_2) - \lambda_2 \sin(2\gamma) = 0 \quad \sin(2\gamma\lambda_3) = 0 \quad (9)$$

where $2\gamma = 2\pi - 2\alpha$ is a function of the notch opening angle. The results of previous equations referred to certain opening angles, namely $2\alpha = 0^\circ, 90^\circ, 120^\circ, \text{ and } 135^\circ$, have been reported in Table 1. It is evident that the reported cases are not the only ones possible in real welded structures; but, on the other hand, the solution of Eq. (9) for a generic value of the notch opening angle would require a numerical algorithm, which could be not available in a programming code or it could be significantly time-consuming to be implemented.

In order to achieve computational implementation of the stress singularity exponents throughout the elaboration tasks developed in ANSYS – PSM, first, parameters λ_i have been calculated by numerically solving Eq. (9) for several values of the notch opening angle 2α ; after that, a n-grade polynomial has been fitted on the obtained results for each parameter λ_i , as shown in Fig. 3. In more detail, the datasets relevant to each parameter λ_i were approximated by means of a polynomial expression, whose grade was recursively adjusted in order to achieve a proper interpolation convergence ($R^2 = 1$). Computational charge was indeed proven to be an influent factor when dealing with parametric calculation within ANSYS® environment. This allowed to embed high-grade polynomial expressions within ANSYS – PSM Macros. The obtained parametric expressions are function of the V-notch opening angle 2α and are summarised by Eq. (10), where coefficients (a, b, c, d, e, f, g) assume different values as a function of the loading mode ($i = 1, 2$ or 3), as highlighted in Table 4.

$$\lambda_i = a \cdot (2\alpha)^6 + b \cdot (2\alpha)^5 + c \cdot (2\alpha)^4 + d \cdot (2\alpha)^3 + e \cdot (2\alpha)^2 + f \cdot (2\alpha) + g \quad \text{where } i = 1, 2, 3 \text{ and } 2\alpha \text{ is in } [^\circ] \quad (10)$$

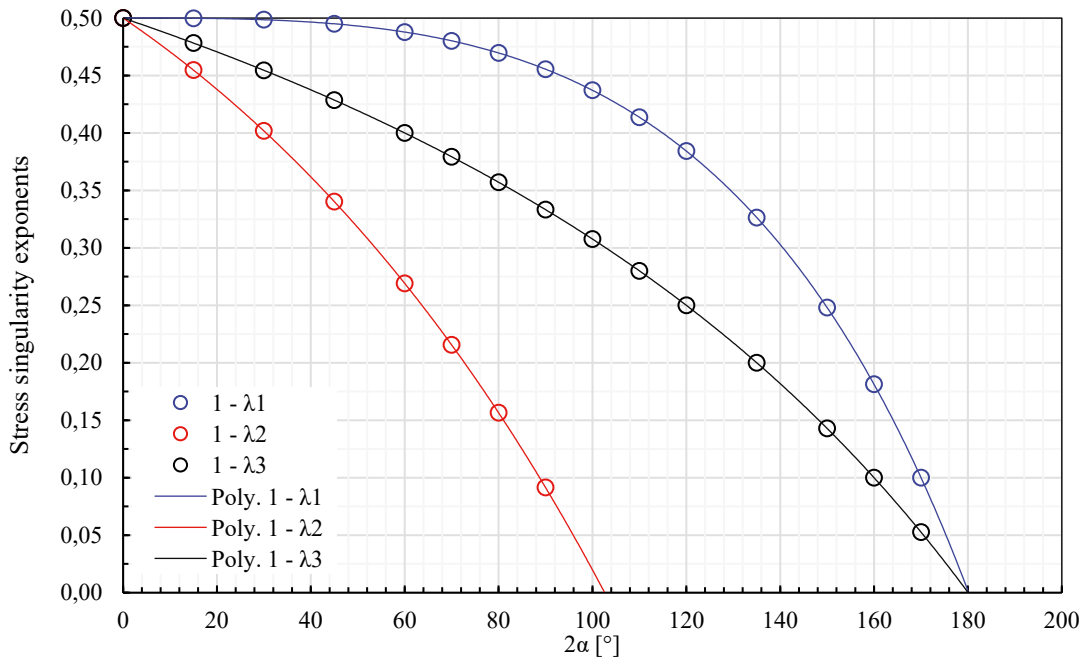


Fig. 3. Stress singularity exponents under mode I, II and III loadings as a function of the notch opening angle 2α

Table 4. Parametric polynomial coefficients to be adopted in Eq. (10) for estimating the stress singularity exponents

| Coefficient | λ_1 | λ_2 | λ_3 |
|-------------|--------------------------|-------------------------|--------------------------|
| a | $1.23958 \cdot 10^{14}$ | 0 | 0 |
| b | $-4.27186 \cdot 10^{12}$ | 0 | $5.26346 \cdot 10^{13}$ |
| c | $8.11426 \cdot 10^{10}$ | $2.37842 \cdot 10^{10}$ | $-8.85746 \cdot 10^{11}$ |
| d | $-3.75746 \cdot 10^9$ | $2.20602 \cdot 10^8$ | $2.30119 \cdot 10^8$ |
| e | $1.77817 \cdot 10^6$ | $1.56491 \cdot 10^5$ | $3.32947 \cdot 10^6$ |
| f | $-1.96363 \cdot 10^5$ | $2.77831 \cdot 10^3$ | $1.39616 \cdot 10^3$ |
| g | 0.5000 | 0.5000 | 0.5000 |

The three polynomial expressions derived for λ_i have been implemented within ANSYS – PSM core, allowing automatic computation of the stress singularity exponents $1 - \lambda_i$ ($i = 1, 2, 3$) as a function of a generic V-notch opening angle 2α .

4.2. Coefficients (ei) for the averaged SED calculation – fitting equations

It has been previously discussed that coefficients e1, e2 and e3 are dependent on the sharp notch geometry and on the material, through the opening angle 2α and the Poisson’s ratio ν (see Table 1). They have been defined by Lazzarin and co-workers (Berto and Lazzarin, 2014; Lazzarin and Zambardi, 2001) dealing with the calculation of the averaged SED, and they can be calculated by the following expressions:

$$e_1 = \frac{\int_{-\gamma}^{\gamma} f_{ij}^I(\theta, \nu) d\theta}{4 \cdot \lambda_1 \cdot \gamma} \quad e_2 = \frac{\int_{-\gamma}^{\gamma} f_{ij}^{II}(\theta, \nu) d\theta}{4 \cdot \lambda_2 \cdot \gamma} \quad e_3 = \frac{(1 + \nu) \cdot \int_{-\gamma}^{\gamma} f_{ij}^{III}(\theta, \nu) d\theta}{6 \cdot \lambda_3 \cdot \gamma} \quad (11)$$

where f_{ij}(θ,ν) represent the angular terms of the strain energy (see (Berto and Lazzarin, 2014; Lazzarin and Zambardi, 2001)), which are not explicitly reported here for sake of brevity. The results of previous equations referred to certain opening angles, i.e. 2α = 0°, 90°, 120°, and 135°, to Poisson’s ratios ν = 0.33 (aluminium alloys) and 0.3 (structural steels) and to a plain strain condition have been reported in Table 1. As already observed for the stress singularity exponents, the values reported in Table 1 do not cover all possible notch geometries and material properties which can be found in real welded structures; but, also in this case, the solution of Eq. (11) for a generic notch opening angle and Poisson’s ratio would require a numerical integration algorithm which should be avoided in programming codes.

In order to provide fitting equations to calculate e1, e2 and e3 which are functions of 2α and ν, two-variables polynomials have been adopted. First, a data-set for each coefficient ei has been generated by numerically solving Eq. (11) for several combinations of the variables 2α and ν. Values of 2α and ν have been considered in the range of interest for the PSM, i.e. opening angles between 0° and 170° and Poisson’s ratios between 0.25 and 0.35. After that, a two-variables interpolation has been performed to derive parametric two-variables polynomial expressions. The obtained expressions define three-dimensional surfaces shown in Fig. 4, providing continuity to both ν and 2α variables within the defined ranges of interest.

The obtained parametric polynomial expressions are function of both V-notch opening angle 2α and Poisson’s ratio ν and are summarised by Eq. (12). The resulting coefficients p_{jk} derive from recursive optimization of the fitting polynomials through grade variation and allow to obtain a percent error between exact (from Eq. (11)) and estimated (from Eq. (12)) values of ei lower than 1%. For each p_{jk} coefficient reported in Eq. (12), j refers to 2α variable’s grade dependency, while k refers to ν variable’s grade dependency.

$$e_i(2\alpha', \nu') = p_{00} + p_{10} \cdot (2\alpha') + p_{01} \cdot (\nu') + p_{20} \cdot (2\alpha')^2 + p_{11} \cdot (2\alpha' \cdot \nu') + p_{02} \cdot (\nu')^2 + p_{30} \cdot (2\alpha')^3 + p_{21} \cdot (2\alpha')^2 \cdot (\nu') + p_{12} \cdot (2\alpha') \cdot (\nu')^2 + p_{40} \cdot (2\alpha')^4 + p_{31} \cdot (2\alpha')^3 \cdot (\nu') + p_{22} \cdot (2\alpha')^2 \cdot (\nu')^2 + p_{50} \cdot (2\alpha')^5 + p_{41} \cdot (2\alpha')^4 \cdot (\nu') + p_{32} \cdot (2\alpha')^3 \cdot (\nu')^2 \quad \text{where } i = 1, 2, 3 \quad (12)$$

where 2α' and ν' are defined by Eq. (13), factors n_{2α} and n_ν and t_{2α} and t_ν being reported in Table 5

$$2\alpha' = \frac{2\alpha - t_{2\alpha}}{n_{2\alpha}} \quad \text{where } 2\alpha \text{ is in } [^\circ] \quad (13a)$$

$$\nu' = \frac{\nu - t_\nu}{n_\nu} \quad (13b)$$

Table 5. Factors to be used in Eq. (13) for 2α and ν interpolation variables.

| Normalisation factor | e1 | e2 | e3 |
|----------------------|---------|---------|----|
| t _{2α} | 89.00 | 48.75 | 0 |
| t _ν | 0.3 | 0.3 | 0 |
| n _{2α} | 51.18 | 30.04 | 1 |
| n _ν | 0.03172 | 0.03180 | 1 |

Table 6. Parametric polynomial coefficients to be adopted in Eq. (12) for estimating SED parameters e_i

| Coefficient | e_1 | e_2 | e_3 |
|-------------|------------------------|------------------------|------------------------|
| p_{00} | $1.465 \cdot 10^{-1}$ | $2.357 \cdot 10^{-1}$ | $3.183 \cdot 10^{-1}$ |
| p_{10} | $-1.684 \cdot 10^{-2}$ | $-5.639 \cdot 10^{-2}$ | $-8.842 \cdot 10^{-4}$ |
| p_{01} | $-8.296 \cdot 10^{-3}$ | $-6.838 \cdot 10^{-4}$ | $3.183 \cdot 10^{-1}$ |
| p_{20} | $-1.816 \cdot 10^{-2}$ | $5.448 \cdot 10^{-3}$ | $8.168 \cdot 10^{-15}$ |
| p_{11} | $3.428 \cdot 10^{-1}$ | $1.556 \cdot 10^{-3}$ | $-8.842 \cdot 10^{-4}$ |
| p_{02} | $-2.868 \cdot 10^{-4}$ | $-1.576 \cdot 10^{-4}$ | 0 |
| p_{30} | $-5.476 \cdot 10^{-4}$ | $-1.068 \cdot 10^{-4}$ | 0 |
| p_{21} | $1.231 \cdot 10^{-3}$ | $-4.998 \cdot 10^{-4}$ | 0 |
| p_{12} | $9.226 \cdot 10^{-5}$ | $7.466 \cdot 10^{-5}$ | 0 |
| p_{40} | $1.451 \cdot 10^{-3}$ | $-8.246 \cdot 10^{-5}$ | 0 |
| p_{31} | $-3.631 \cdot 10^{-4}$ | $6.646 \cdot 10^{-5}$ | 0 |
| p_{22} | $3.024 \cdot 10^{-5}$ | $-1.595 \cdot 10^{-5}$ | 0 |
| p_{50} | $1.547 \cdot 10^{-4}$ | 0 | 0 |
| p_{41} | $-1.329 \cdot 10^{-4}$ | 0 | 0 |
| p_{32} | $-7.563 \cdot 10^{-6}$ | 0 | 0 |

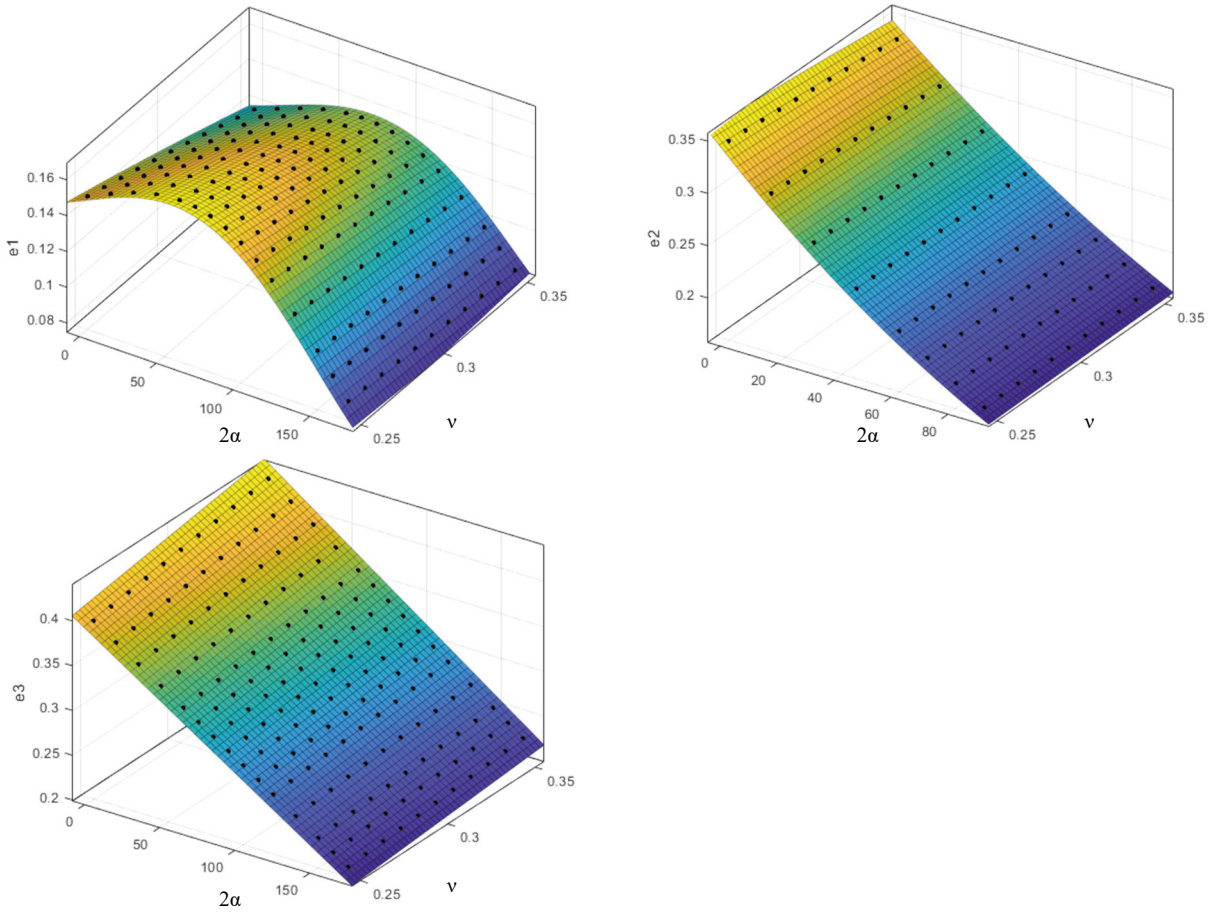


Fig. 4 Surface plot of polynomial parametric interpolants of averaged SED coefficients.

Finally, the p_{ij} coefficients assume different values as a function of the loading mode ($i = 1, 2$ or 3), as highlighted in Table 6.

The three polynomial expressions derived for SED coefficients e_i have been implemented within ANSYS – PSM core, allowing automatic computation of the coefficients e_i as a function of generic V-notch opening angle 2α and Poisson's ratio ν

5. ANSYS – PSM subroutine for ANSYS Mechanical APDL

The concept idea behind ANSYS – PSM was to create an interactive extension within ANSYS® Mechanical APDL post-processing environment, in order to provide the analyst a tool for the automatic fatigue strength assessment of a generic arc-welded structure, which takes advantage of the Peak Stress Method. ANSYS – PSM core retraces the whole method's workflow shown in Fig. 2, automatically performing the following operations, which the FE analyst should manually complete instead:

- performs compatibility checks between the FE model under analysis and the PSM requirements (summarised in Table 2), i.e. adopted element type and mesh density ratio a/d . A check of the mesh pattern, i.e. the number of elements shared by nodes at V-notch tip, is also performed, whenever the adopted element type requires specific requirements according to PSM.
- identifies the material adopted in the FE analysis.
- identifies actual V-notches among all geometrical features (lines) of either 2D or 3D FE models.
- evaluates the V-notch opening angle at either weld toe or root sides.
- defines, locates and manages local coordinate systems and to retrieve the peak stress components at either weld toe or root sides.
- computes PSM-related parameters, i.e. stress singularity exponents and SED coefficients, taking advantage of fitting equations (10) and (12), as a function of the model's local geometry and material.
- combines peak stress components into an equivalent peak stress and evaluate the local biaxiality ratio.
- estimates fatigue life on each detected V-notch site of the FE model, referring to the proper PSM fatigue design curve.

5.1. Development of ANSYS – PSM: programmer's point of view

ANSYS – PSM structure is composed of two stages: a front-end interactive GUI (Graphical User Interface) and a back-end Software Core. The UIDL GUI is designed to be the bridge between the analyst and the FE code routines, while the Software Core, coded in both APDL and UIDL languages, is programmed to define the computational workflow of the automated PSM analysis. ANSYS – PSM architecture is grounded on three *levels*, each one relating to a dedicated UIDL graphical Menu, all together building the interactive front-end GUI. Each *level* connects to the Software Core through specific gateways, allowing back-end execution of the *level*-relative setup, analysis or post-processing tasks through dedicated APDL Macros. The ANSYS – PSM analysis workflow according to a programmer's point of view is sketched in Fig. 5 and will be described in the following.

The first *level* relates to ANSYS – PSM Setup Phase and is associated to a UIDL General Setup Menu. This start-up Menu is designed in order to collect, all at once, essential information and user inputs which ANSYS – PSM will subsequently store into ANSYS® static parameters and use alongside PSM workflow's implementation. All parameters defined and calculated throughout ANSYS – PSM workflow are stored within ANSYS® Parameters Suite, being steadily accessible by the Software Core. UIDL graphical features have been designed within the General Setup Menu, in order to provide the Software Core access points to collect information about the model's element type properties, material properties, geometrical parameters, loading conditions' scenario and fatigue design settings. Start-up APDL Macros, which are automatically executed throughout the Setup Phase, allow the Software Core to perform specific checks:

- Pre-Checks: provide the Setup Menu general data related to the model's material, mesh and solution properties. Setup Pre-Checks are performed through specific APDL Macros, called *onrefresh* of the General Setup Menu.

- Post-Checks: performed whenever the General Setup Menu is finalized. Taking advantage of information stored throughout the Setup Phase, three main Compatibility Checks are performed by the Software Core, through dedicated APDL Macros, called *onupdate* of the General Setup Menu:
 - Material Properties are checked in order to evaluate parameters designed to guide the Software Core on the selection of the proper PSM fatigue design curve.
 - Mesh Compatibility is checked in terms of the minimum mesh density ratio a/d . The associated APDL Macro is programmed to raise a message prompt to the user, providing information about the model's Mesh Status.
 - Model Compatibility is checked in terms of model's geometrical characteristics and minimum welded plates' thickness.

The second *level* relates to ANSYS – PSM Analysis Phase. This *level* is associated to a UIDL Analysis Setup Menu, designed to firstly provide the user information on the analysis *time-to-completion*. This Analysis Setup Menu is also designed to accept an input by the user through a programmed *radio* button, giving him full control on the selection of the Analysis Method, either full-automatic or interactive. *Onrefresh* of the Analysis Setup Menu, an APDL Macro is executed, performing Analysis Pre-Checks to evaluate eventual model-related boundary and loading conditions (e.g. symmetry boundary conditions). *Onupdate* of the Analysis Setup Menu, the APDL-driven Fatigue Analysis routine is sparked. The routine is designed to loop on model's lines, executing a series of checks and local geometrical elaborations aimed at retrieving the peak stress data required by the PSM calculation workflow. The first step of the automated routine consists of the V-notch identification and local analysis. Each line of the model is considered at a time and a scheduled elaboration is automatically performed by the Software Core, in order to evaluate the model's geometry in proximity of the considered line, eventually defining it as a V-notch site. Once a model's line is recognized as a V-notch tip profile, the Software Core proceeds to a nodal-level analysis, involving all nodes belonging to that line. This level of the analysis corresponds to the application of the PSM to the model's nodes and is driven by a looping automated APDL algorithm, aimed at automatically performing the tasks sketched in Fig. 2.

On the other side, if the considered line is not recognized as a V-notch tip profile, the nodal-level analysis is by-passed by the Software Core, which automatically steps from line i to line $i+1$ on the current V-notch analysis loop. All retrieved data is stored by the Software Core in multiple ways. The stress and life values are collected in ANSYS® arrays parameters and tables (within ANSYS® Parameters Suite), in order to be steadily available alongside the further post-processing phase. Moreover, estimated life results are formatted and saved through APDL algorithms into dedicated external *.lis* files and registered to ANSYS – PSM Analysis Log. Analysis Log is a section of the program directly accessible by the user through the ANSYS – PSM GUI, aimed at collecting information and results coming from the analysis, providing a reference log throughout which the user can access ANSYS – PSM outcomes at any time.

Second-*level* analysis phase automatically initializes the third-*level* post-processing phase by calling out the 'life contour plot' function. The associated APDL sub-routine allows to generate a coloured line contour plot aimed at providing the user a graphical presentation of estimated fatigue life distributions along detected V-notch lines. The third *level* relates to ANSYS – PSM results elaboration and post-processing phase. This *level* is defined by a UIDL Post-Processing Menu, designed in order to provide the user a cluster of options and features aimed at results elaboration and visualization. Alongside 'life contour plot' UIDL shortcut, 'query life results' UIDL shortcut enables a post-processing function designed in the same way as the ANSYS® native 'query results' feature: an integrated UIDL/APDL algorithm allows the user to visualize estimated life values on each node along detected V-notch lines by manually clicking on the desired node site. 'Clear life results' UIDL shortcut finalizes the on-going analysis by back-calling an APDL clearing algorithm, programmed to delete all results and cache generated during last PSM fatigue analysis.

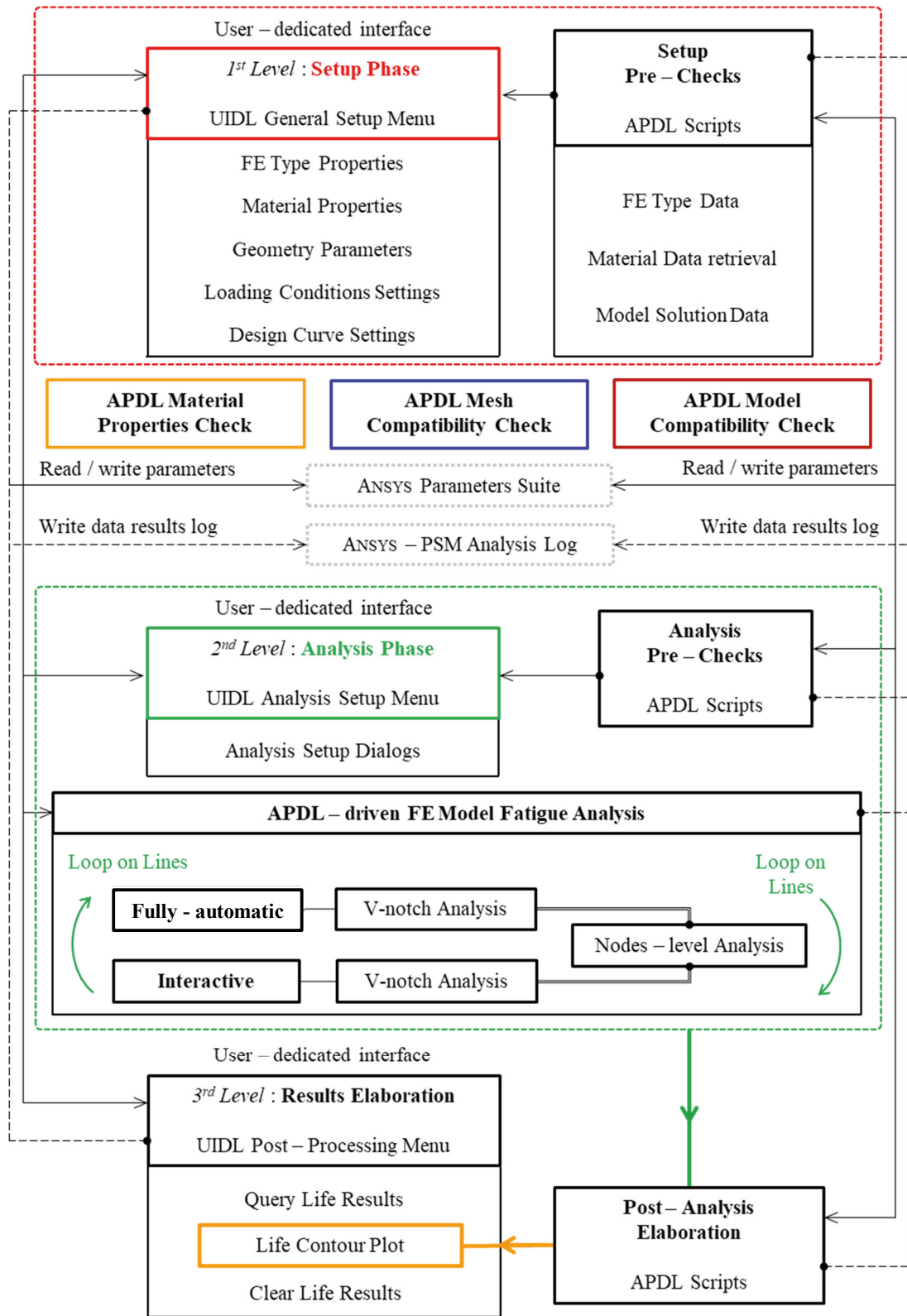


Fig. 5 ANSYS – PSM analysis workflow: programmer’s point of view

5.2. ANSYS – PSM analysis workflow: FE analyst's point of view

As described in previous section, ANSYS – PSM subroutine is designed to work on three *levels*. Each level is associated to a specific phase of the full analysis' workflow and is characterized by the integration of graphical interactive features and software algorithms. In order to build a gateway between the analyst and the former computational soul of ANSYS – PSM, a dedicated GUI (Graphical User Interface) has been developed. ANSYS – PSM GUI is designed to be interactive and to grant the analyst a full control of the back-end software counterpart. The ANSYS – PSM analysis workflow according to a FE analyst's point of view is sketched in Fig. 6 and will be described in the following.

To perform a PSM-based fatigue assessment of a welded structure, ANSYS – PSM requires to start from a solved FE model as input. More in detail:

- The 2D or 3D model should be created or imported within ANSYS® Mechanical APDL environment, including both the geometry of the structure and also the weldments (see example in Fig. 2).
- The analyst is requested to generate a FE mesh, which must respect the PSM requirements, in terms of supported element type and mesh density ratio (a/d), taking advantage of ANSYS® free mesh generation algorithm.
- The model must be loaded and constrained, and, after that, solved in order for ANSYS – PSM to access solution data and results.

Once the model is properly solved, ANSYS – PSM utilities can be accessed and launched directly from ANSYS® Main Menu, through the ANSYS – PSM Suite voice. ANSYS – PSM Setup can be accessed directly from the shortcut 'ANSYS – PSM Setup' within ANSYS – PSM Suite, revealing the main setup window. Throughout the window, the analyst is requested to input values and define desired settings by selection, i.e. geometry setup settings and fatigue design setting. These setup options are strictly necessary in order for ANSYS – PSM to work properly. Among requested analyst inputs and selections are the following:

- Input the characteristic dimension a , which characterise the size of the weld toe or root feature of the analysed welded structure.
- Input the local element size d adopted to generate the FE mesh.
- Input the minimum welded plates' thickness T_{\min} of the entire structure.
- Select between the different PSM-calibrated fatigue design curves (i.e. structural steel or aluminium alloys).

Once all settings have been checked and/or defined, the analyst can finalize the setup phase by clicking the OK button. ANSYS – PSM program is now called to process and check all information and data declared throughout the setup phase. A first post-setup check is carried out on material's properties: ANSYS – PSM will automatically extract model's material properties, i.e. Young's modulus and Poisson's ratio, checking compatibility with respect to the target fatigue design curve selected by the analyst in terms of material type. A second check is performed on model's mesh compatibility, in terms of minimum mesh density ratio a/d , automatically calculated and evaluated by the Software Core. As all setup Post-Checks are completed, ANSYS – PSM prompts the analyst with a mesh status dialog, i.e. a window providing information about the model's mesh compatibility, in terms of mesh density ratio as compared to the required minimum ones for each loading mode (see Table 2). A third check is finally carried out on model compatibility, in terms of declared minimum welded plate's thickness with respect to PSM requirements (see Table 3). These checks are design to act as a breakwater, allowing the analyst to proceed to the next analysis steps only whenever full compatibility is confirmed between the model under exam and the PSM requirements. Whenever even one of these checks fails, the analyst will be reverted to the starting point and prompted with warning dialogs and advices on how to adjust the model's current status, in order to grant it compatibility with the specific failed requirements.

Once the model's compatibility is verified, the analyst is allowed to proceed to the Analysis Phase. ANSYS – PSM analysis phase can be directly accessed from the shortcut 'Start ANSYS – PSM' within ANSYS – PSM Suite. The analyst is now prompted with a pre-analysis setup menu, designed to collect analyst's selection on the desired analysis method. ANSYS – PSM is programmed to offer the following two analysis methods.

- Full automatic analysis. This method is designed to provide ANSYS – PSM full-control on the analysis phase. ANSYS – PSM is therefore in charge of analysing the entire FE model, line by line, identifying V-notch lines and performing PSM-related calculation on each detected V-notch tip profile.

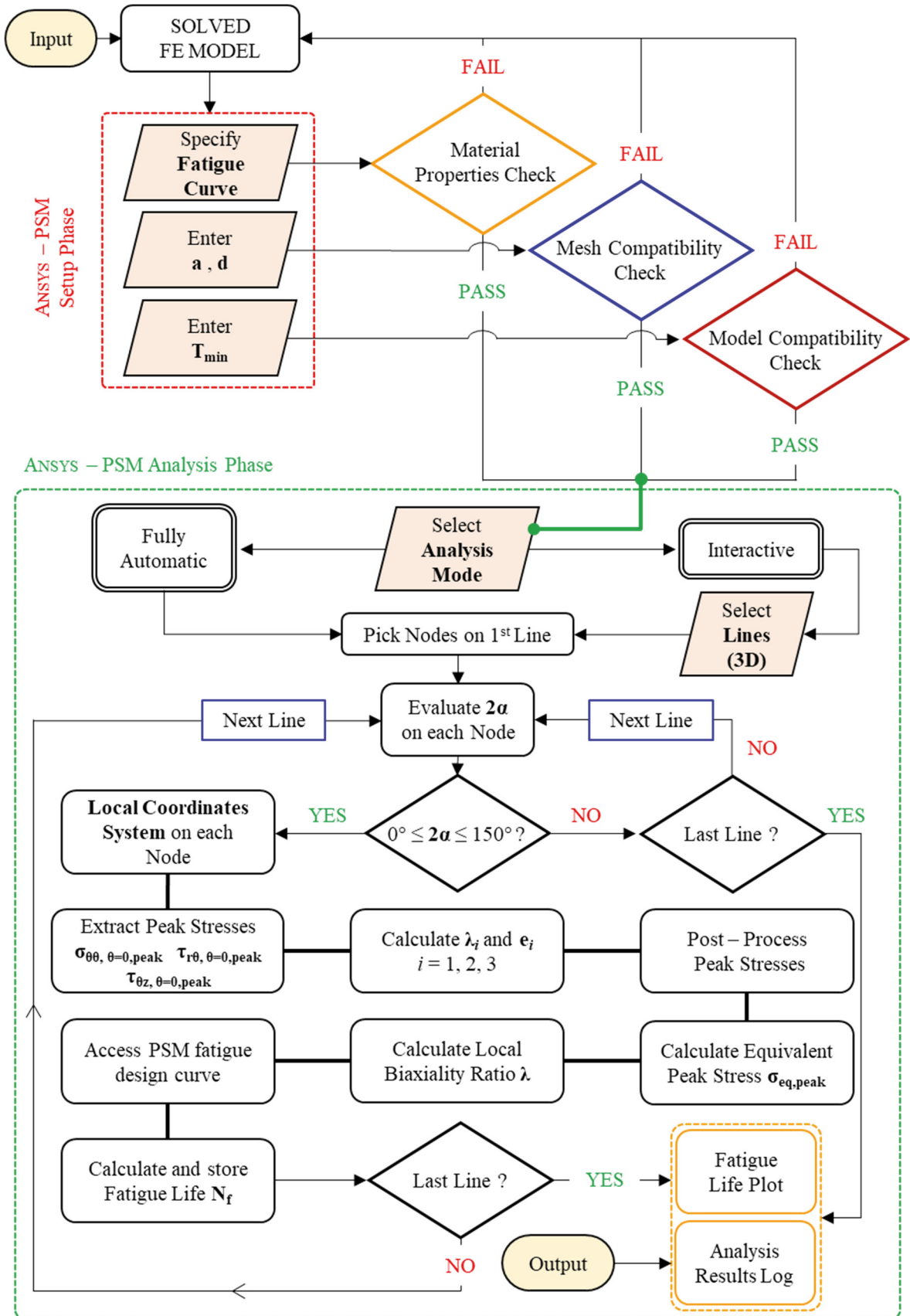
- Interactive analysis. This method is designed to allow the analyst to select one or more specific lines on which the analysis will be carried out. Once the analyst selected the desired model's lines, ANSYS – PSM is programmed to regain control of the analysis process, to evaluate selected lines as V-notch sites and perform PSM-related calculations on each detected V-notch tip-profile.

ANSYS – PSM gains full-control of the Analysis workflow regardless of the analysis method, once a defined set of model's lines is passed to the program (either all model's lines in the case of a 'full automatic analysis' or a set of user-selected lines in the case of an 'interactive analysis'). A looping section is started by ANSYS – PSM on selecting the first model's line of the set. The analysis phase is carried out through the following steps:

- a) all nodes belonging to the considered line are selected and V-notch opening angle is locally evaluated on each nodal position. The V-notch identification sub-routine perform the following tasks:
 - evaluates the local orientation of the considered line;
 - defines construction circular areas at each nodal location along the considered line, each area being locally orthogonal to the line's tangent direction;
 - intersects the model's volume with the construction circular areas;
 - evaluates the material-side angles subtended by the resulting local circular sectors;
 - calculates the actual air-side V-notch opening angles from the conjugate material-side angles: only if the air-side opening angle is in the range between 0° and 150° , the line is considered as a V-notch tip profile.
- b) once the line is recognized as a V-notch tip profile, a local coordinate system is automatically defined and orientated on each nodal location. The automated algorithms allow to locally align z -direction (Z in Ansys) tangent to the notch tip profile, θ -direction (Y in Ansys) originating from the notch bisector line and r (X in Ansys) being the radial direction, according to the PSM requirements.
- c) peak stress components $\sigma_{\theta\theta,\theta=0,\text{peak}}$, $\tau_{r\theta,\theta=0,\text{peak}}$ and $\tau_{\theta z,\theta=0,\text{peak}}$ are evaluated node by node referring to the local coordinate system and stored in dedicated ANSYS® Arrays.
- d) in the case of FE meshes defined by adopting 10-nodes tetra or 4-nodes tetra elements, the calculated peak stress components are automatically elaborated into averaged peak stresses, according to Eq. (5).
- e) PSM-related parameters, i.e. stress singularity exponents and SED coefficients, are computed taking advantage of fitting equations (10) and (12), as a function of the model's local geometry and material. After that parameters f_{wi} are calculated at each nodal location by means of Eq. (7).
- f) peak stress values are combined into a local equivalent peak stress at each nodal location of the selected line, by mean of Eq. (6).
- g) local biaxiality ratio λ is evaluated on each node along considered model's line by Eq. (8). A PSM fatigue design curve is univocally selected and addressed by ANSYS – PSM according to the local biaxiality ratio value and the minimum welded plates' thickness (see Table 3).
- h) a fatigue life N_f value is estimated on each analysed node belonging to the selected line. Fatigue life values estimated on analysed nodes are stored either within ANSYS – PSM Analysis Log, either within dedicated external *.lis* files, in order to be steadily available to further post-processing elaborations and results visualization.

Once the selected line has been completely analysed, ANSYS – PSM loops to the following model's line on the passed set, performing steps (a – h) from the beginning on the new line. Whether a line is not recognized as a V-notch site, steps (b – h) are skipped and the analysis moves to the straight-following line. The entire full-automated part of the Analysis Phase is tracked by means of an Analysis Progress Bar, designed in order to visualize progress increments in terms of number of currently analysed lines over all selected model's lines. The Progress Bar is constantly visualized and updated by ANSYS – PSM and can be referenced by the analyst in order to estimate the process completion status and *time-to-completion* throughout the entire Analysis.

Once all model's lines have been analysed, ANSYS – PSM finalizes the Analysis Phase and enters post-processing phase in order to provide results elaboration and visualization. Fatigue analysis results are visualized in terms of graphical line contour plots displaying estimated fatigue life distributions along analysed V-notch tip profile. The Life Contour Plot is automatically elaborated and visualized on the screen at the end of the Analysis Phase and is designed to be consulted through a color scale, normalized with respect to the maximum detected fatigue life value throughout the entire model's structure. The color bands are designed to range from red color, corresponding to the minimum fatigue life value, to blue color, corresponding to the maximum fatigue life value.



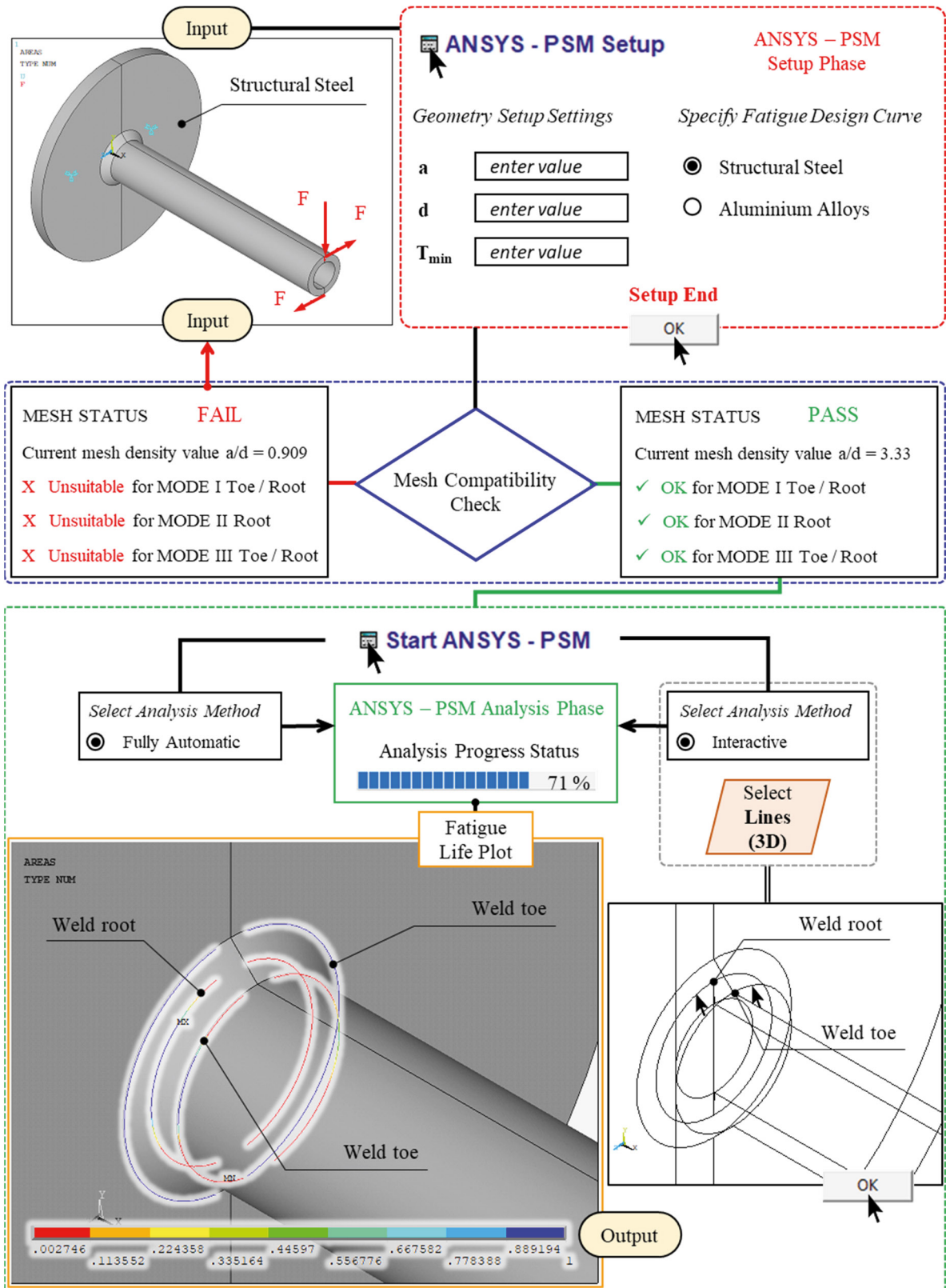


Fig. 6 ANSYS - PSM analysis workflow: FE analyst's point of view

Conclusions

The Peak Stress Method (PSM) employs the singular, linear elastic peak stresses evaluated at the weld toe and weld root by means of FE analyses with coarse meshes to rapidly estimate the mode I, II and III Notch Stress Intensity Factors. By using the PSM in combination with the averaged Strain Energy Density (SED) fatigue strength criterion, a so-called equivalent peak stress has been defined to assess either weld toe or weld root fatigue failures in conjunction with a properly calibrated design curve. In the present contribution, a subroutine, named ANSYS-PSM, has been developed in the post-processing environment of ANSYS® FE code to automate the application of the PSM to 2D or 3D FE models. More in detail, the subroutine automatically allows: (i) to identify the notch opening angle at weld toe or weld root; (ii) to check if the PSM requirements, i.e. element type, mesh pattern and minimum mesh size, have been respected; (iii) to define a local coordinate system having axes properly aligned; (iv) to derive the peak stresses in the local reference system and to combine them to calculate the equivalent peak stress; (v) to identify the proper fatigue design curve to be employed; (vi) to present graphically the most critical location and to estimate the fatigue life of the analysed welded structure. Because of the coarse FE analyses required and the automated procedure to estimate the fatigue life, the PSM-application might be useful in the everyday design practice, also in the case when large-scale structures and complex loading conditions are considered.

References

- Berto, F., Lazzarin, P., 2014. Recent developments in brittle and quasi-brittle failure assessment of engineering materials by means of local approaches. *Mater. Sci. Eng. R Reports* 75, 1–48. doi:10.1016/j.mser.2013.11.001
- Campagnolo, A., Meneghetti, G., 2018. Rapid estimation of notch stress intensity factors in 3D large-scale welded structures using the peak stress method. *MATEC Web Conf.* 165. doi:10.1051/mateconf/201816517004
- Campagnolo, A., Meneghetti, G., Babini, V., Riboli, M., Spagnoli, A., 2019a. Multiaxial fatigue assessment of welded steel details according to the peak stress method based on tetra elements. *MATEC Web Conf.* 300, 19002. doi:10.1051/mateconf/201930019002
- Campagnolo, A., Roveda, I., Meneghetti, G., 2019b. The Peak Stress Method combined with 3D finite element models to assess the fatigue strength of complex welded structures. *Procedia Struct. Integr.* 19, 617–626. doi:10.1016/j.prostr.2019.12.067
- Eurocode 3: Design of steel structures – part 1–9: Fatigue, 2005. . CEN.
- Gross, B., Mendelson, A., 1972. Plane elastostatic analysis of V-notched plates. *Int. J. Fract. Mech.* 8, 267–276. doi:10.1007/BF00186126
- Lazzarin, P., Lassen, T., Livieri, P., 2003. A notch stress intensity approach applied to fatigue life predictions of welded joints with different local toe geometry. *Fatigue Fract. Eng. Mater. Struct.* 26, 49–58. doi:10.1046/j.1460-2695.2003.00586.x
- Lazzarin, P., Livieri, P., Berto, F., Zappalorto, M., 2008. Local strain energy density and fatigue strength of welded joints under uniaxial and multiaxial loading. *Eng. Fract. Mech.* 75, 1875–1889. doi:10.1016/j.engfracmech.2006.10.019
- Lazzarin, P., Sonsino, C.M., Zambardi, R., 2004. A notch stress intensity approach to assess the multiaxial fatigue strength of welded tube-to-flange joints subjected to combined loadings. *Fatigue Fract. Eng. Mater. Struct.* 27, 127–140. doi:10.1111/j.1460-2695.2004.00733.x
- Lazzarin, P., Tovo, R., 1998. A notch intensity factor approach to the stress analysis of welds. *Fatigue Fract. Eng. Mater. Struct.* 21, 1089–1103. doi:10.1046/j.1460-2695.1998.00097.x
- Lazzarin, P., Zambardi, R., 2001. A finite-volume-energy based approach to predict the static and fatigue behavior of components with sharp V-shaped notches. *Int. J. Fract.* 112, 275–298. doi:10.1023/A:1013595930617
- Livieri, P., Lazzarin, P., 2005. Fatigue strength of steel and aluminium welded joints based on generalised stress intensity factors and local strain energy values. *Int. J. Fract.* 133, 247–276. doi:10.1007/s10704-005-4043-3
- Meneghetti, G., 2013. The peak stress method for fatigue strength assessment of tube-to-flange welded joints under torsion loading. *Weld. World* 57, 265–275. doi:10.1007/s40194-013-0022-x
- Meneghetti, G., 2012. The use of peak stresses for fatigue strength assessments of welded lap joints and cover plates with toe and root failures. *Eng. Fract. Mech.* 89, 40–51. doi:10.1016/j.engfracmech.2012.04.007
- Meneghetti, G., Campagnolo, A., 2020. State-of-the-art review of peak stress method for fatigue strength assessment of welded joints. *Int. J. Fatigue* 139, 105705. doi:10.1016/j.ijfatigue.2020.105705
- Meneghetti, G., Campagnolo, A., Avalle, M., Castagnetti, D., Colussi, M., Corigliano, P., De Agostinis, M., Dragoni, E., Fontanari, V., Frendo, F., Goglio, L., Marannano, G., Marulo, G., Moroni, F., Pantano, A., Rebora, A., Scattina, A., Spaggiari, A., Zuccarello, B., 2018. Rapid evaluation of notch stress intensity factors using the peak stress method: Comparison of commercial finite element codes for a range of mesh patterns. *Fatigue Fract. Eng. Mater. Struct.* 41. doi:10.1111/ffe.12751
- Meneghetti, G., Campagnolo, A., Babini, V., Riboli, M., Spagnoli, A., 2019. Multiaxial fatigue assessment of welded steel details according to the

- peak stress method: Industrial case studies. *Int. J. Fatigue* 125, 362–380. doi:10.1016/j.ijfatigue.2019.04.014
- Meneghetti, G., Campagnolo, A., Berto, F., 2015. Fatigue strength assessment of partial and full-penetration steel and aluminium butt-welded joints according to the peak stress method. *Fatigue Fract. Eng. Mater. Struct.* 38, 1419–1431. doi:10.1111/ffe.12342
- Meneghetti, G., Campagnolo, A., Rigon, D., 2017a. Multiaxial fatigue strength assessment of welded joints using the Peak Stress Method – Part I: Approach and application to aluminium joints. *Int. J. Fatigue* 101, 328–342. doi:10.1016/j.ijfatigue.2017.03.038
- Meneghetti, G., Campagnolo, A., Rigon, D., 2017b. Multiaxial fatigue strength assessment of welded joints using the Peak Stress Method – Part II: Application to structural steel joints. *Int. J. Fatigue* 101, 343–362. doi:10.1016/j.ijfatigue.2017.03.039
- Meneghetti, G., Guzzella, C., 2014. The peak stress method to estimate the mode I notch stress intensity factor in welded joints using three-dimensional finite element models. *Eng. Fract. Mech.* 115, 154–171. doi:10.1016/j.engfracmech.2013.11.002
- Meneghetti, G., Lazzarin, P., 2007. Significance of the elastic peak stress evaluated by FE analyses at the point of singularity of sharp V-notched components. *Fatigue Fract. Eng. Mater. Struct.* 30, 95–106. doi:10.1111/j.1460-2695.2006.01084.x
- Qian, J., Hasebe, N., 1997. Property of eigenvalues and eigenfunctions for an interface V-notch in antiplane elasticity. *Eng. Fract. Mech.* 56, 729–734. doi:10.1016/S0013-7944(97)00004-0
- Radaj, D., Sonsino, C.M., Fricke, W., 2006. *Fatigue Assessment of Welded Joints by Local Approaches*, 2nd ed. Woodhead Publishing, Cambridge.
- Visentin, A., 2020. *Automated Fatigue Strength Assessment of V - Notch Structures according to the Peak Stress Method*. University of Padova.
- Williams, M.L., 1952. Stress singularities resulting from various boundary conditions in angular corners of plates in tension. *J Appl Mech* 19, 526–528.