Swiss delegation

IIW-doc. XIII-2650-16 rev-1

Round robin on local stress evaluation for fatigue by various FEM software

A. Nussbaumer, V. Grigoriou

Swiss federal institute of technology in Lausanne, EPFL
Resilient steel structures laboratory, RESSLab

Station 18, 1015 Lausanne
Switzerland

Abstract

The objective of this Round Robin is to assess the software influence on the structural/local stress values as determined by means of various methods. In particular the influence of (a) FEM software solver, (b) potentially software-depending element characteristics (e.g. exact shape functions) and (c) software-depending post processing procedures (e.g. extrapolation from integration points and stress averaging methods. Two different typical problems are studied, one 2D geometry and one 3D geometry, using FE models with various element types and varying mesh finesse. For each of the models, the participants receive files with node coordinates and element numbering, in order to eliminate variations in mesh geometry. Results calculated by the Round Robin participants using various software are reported for structural/local stress values defined according to the following methods: hot-spot, Xiao and Yamada, Dong, Haibach, critical distance and equivalent notch stress. Eventually, these two problems could be proposed as benchmark examples for use by engineers who are not familiar with these methods in the validation of FE modeling procedures aiming at structural/local stress evaluation.

Key-words: fatigue, structural stress, finite elements, software, round robin

1. MOTIVATION AND SCOPE

Many studies have been made to compare different methods of structural/local stress calculation for different structural details in the framework of fatigue life evaluation [1-5]. These included comparison between some or all of the following methods, which can be regrouped in 4 families:

- Structural stress methods using surface stress extrapolation:
  - hot spot methods [6];
- Structural stress methods based on the stress at a single point:
  - Xiao-Yamada's method [7],
  - Haibach's method [8];
- Through-thickness linearization methods:
  - Dong's method [9];
- Local stress methods:
  - Critical distance method [10],
  - Local effective notch stress [11].

Usually, the same software was used to generate all models. The influence of mesh size was studied. In a few studies, different research groups used different software [4] but the influence of the software itself on the obtained results was never assessed.

The objective of this Round Robin is to assess this influence; that is in particular the influence of:

- FEM software solver and software element (shape function, stress-strain integration technics, stress extrapolation at the nodes etc.),
- Software nodal averaging vs point value solutions.

The scope of the study is limited to the software influence on the local stress determination using different methods. This influence is evaluated by means of a round robin in which the same series of stress analysis problems are solved independently by the participants using various Finite Element (FE) software.

2. METHODOLOGY

Two different typical problems, one two-dimensional (2D) (plane elements) and one three-dimensional (3D) (solid elements), are selected for analysis using models with different element types and varying mesh refinement. In total five different models are specified for the 2D problem and three for the 3D problem. These models combined with the ("appropriate" for each model) structural/local stress evaluation methods listed above result in a total of 19 and 9 evaluation cases for the 2D and the 3D problem respectively; i.e. 28 values to be reported by each participant. These values correspond to stress concentration factors (SCF) calculated assuming unit nominal (remote) stress applied as external load. The study problems and the evaluation cases are presented in detail in Section 3.

Since the objective is to assess the software influence itself, the basic modeling parameters including geometric form and dimensions, loading configuration, material properties, discretization pattern, element types and stress-strain integration scheme (full or reduced) should be fixed as precisely as possible. Also, the stress invariant to be used (1st principal stress) and the locations at which it should be evaluated for the application of each of the above structural/local stress evaluations methods should be precisely defined. The prescribed basic modeling parameters are also presented in detail in Section 3.

On the other hand, the decision was taken not to give specific instructions concerning the parameters related to the details of the algorithms which are employed, for instance, for external force lumping at nodes, stress-strain integration, stiffness matrix inversion, stress extrapolation at nodes, nodal averaging etc. The reason for this decision is that the options for these parameters depend largely on the software.

The detailed description of the study problems including the prescribed basic modeling parameters were formulated in a document which was sent to the participants. In order to eliminate variations due to the mesh geometry, the participants also received files with node coordinates and element numbering for each of the studied models. The participants were asked to provide their results using for each evaluation case a separate template report sheets summarizing the basic modeling parameters. When most of the participants have provided their results a series of two web meetings were organized in order to examine whether a number of observed discrepancies were due to software or to an alternative interpretation of the instructions.

Until the moment this report is written, results have been received from the participants listed in Table 1 along with the software used by each of them.

Table 1: Round robin participants and used software

1	A. Nussbaumer	Swiss Federal Institute of Technology in Lausanne, EPFL (CH)	a	ABAQUS 6.11
			b	ANSYS 16.2
2	M. Stoschka	University of Leoban (DE)	a	ABAQUS 6.12
			b	Optistruct 13.0.0.98
			c	Comsol 4.4
3	J. Maljaars, A. Slobbe	Netherlands Organization for Applied Scientific Research, TNO (NL)		DIANA 9.6
4	G. Meneghetti, A Campagnolo	Universit of Padova, UNIPD (IT)		ANSYS 14
5	M. Marzin, F. Lefebvre	Technical Center for Mechanical Industries, CETIM (FR)		ABAQUS 6.13
6	L. Borges	Structurame		Code_Aster

3. STUDY CASES

3.1. Geometry and material

One 2D and one 3D geometric forms, corresponding to two common welding configurations, are analyzed in the framework of this study. The 2D geometric form, with dimensions shown in Fig. 1(a), corresponds to a cruciform joint which is one the most widely used and modeled joints. The 3D geometric form, with dimensions shown in Fig. 1(b), corresponds to a longitudinal attachment.

Both the above forms present considerable advantages: possibility to compare with previous studies (e.g. [2, 3]) and with experimental results, use of symmetries in the modeling which reduce computation cost, presence of a weld toe initiation site and, if wanted in subsequent study, of a weld root initiation site. However, in the present study, the forms do not have internal boundaries which implies that they correspond to the case of full penetration welds. In addition, the 3D form presents a significant 3D effect of the stress distribution in the stress concentration region.

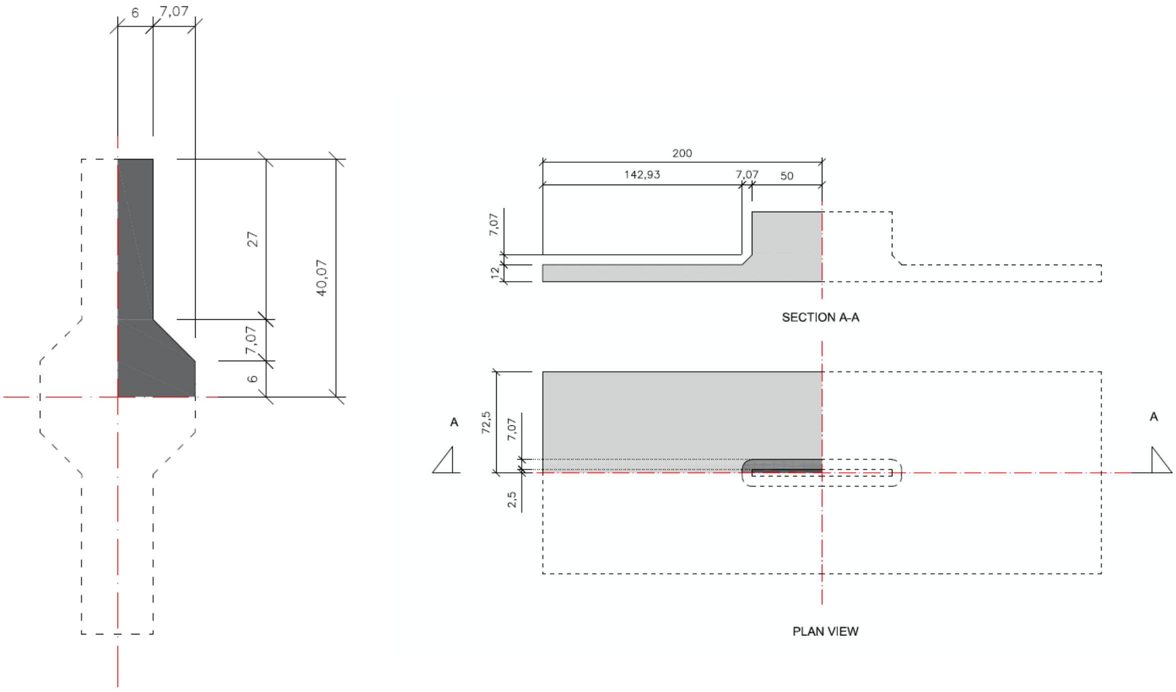
The used symmetry planes are shown in Fig. 1. It should be noticed that for the 2D form, only half the thicknesses of both the loaded and the transvers elements are included in the FE model; while for the 3D form the base plate is modeled with its total thickness, while the attachment is modeled with half its thickness.

For the application of the notch stress method the above geometric forms are slightly modified to include filets at the toes of the welds the radius of these fillets is taken equal to 1 mm according to [11].

Only one material is considered: mild steel with Young's modulus E = 210000 MPa and Poisson ratio ν = 0.3.

3.2. Finite element discretization

The 2D form is discretized using three types of isoperimetric elements: linear triangular (3 corner nodes), quadratic triangular (3 corner nodes + 3 mid side nodes at and quadratic quadrilateral elements, and two mesh densities: Coarse mesh (relatively fine according to [12]) and fine mesh. These meshes are depicted in Fig. 2a along with the element dimensions in the stress concentration region. In addition, "very fine" 2D meshes (0.05 mm element size, without radius), intended for calculations of the local stress according to the critical distance method (see below) is proposed after the feedback received during the presentation of this work in the 2016 IIW annual meeting in Melbourne. This meshes are depicted in Fig. 2b. However, at the time this revision is written, only two participants have submitted results for this additional "very fine" meshes.



(a) (b)

Fig. 1: Geometry of the study cases: (a) 2D problem, (b) 3D problem.

3.3. Loading and boundary conditions

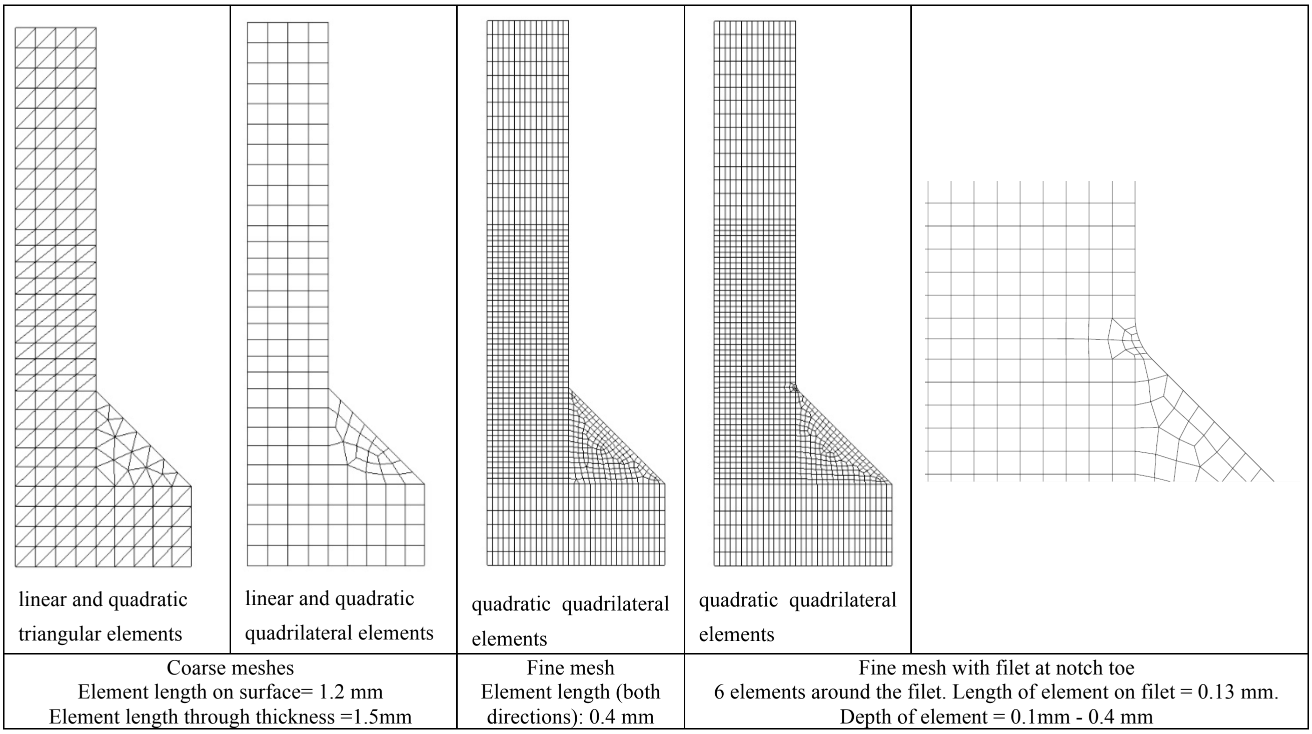
For the 2D models, positive (tensile) uniform unit line load should be applied along line A-A as shown in Fig. 4(a). The translational degrees of freedom (DOF) in the global X axis (Ux) are set to 0 for all the nodes located on Line C-C (mirror symmetry about Y axis), the translational DOFs in the global Y axis (Uy) are set to 0 for all the nodes located on line B-B (mirror symmetry about X axis).

For the 3D models a positive uniform surface load is applied on Face A of the model (Fig. 4(b)). Ux DOFs are set to zero for all node located on Face B while Uz DOFs are set to zero for all nodes on Face C. The model has no symmetry with respect to plane XY.

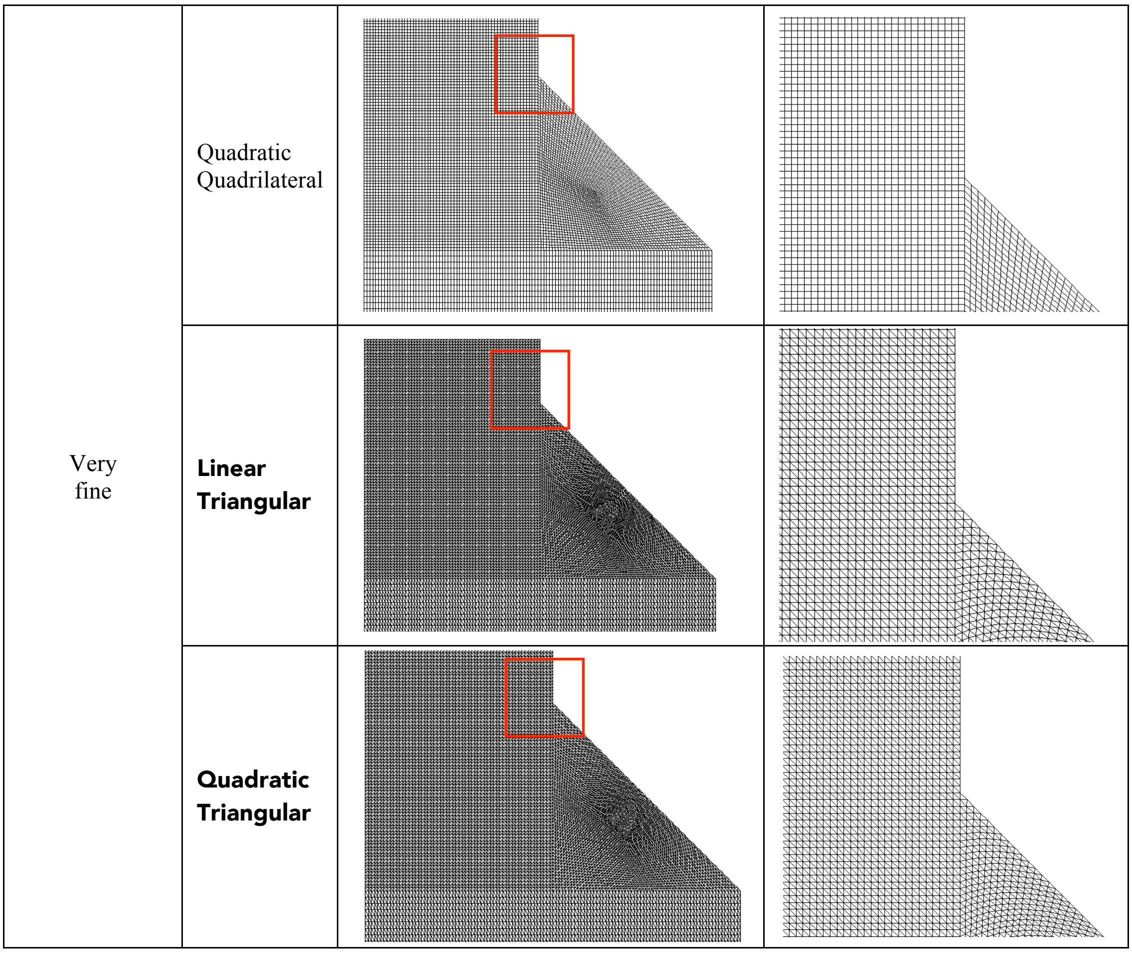
The above nominal stress loads, can be transformed into equivalent nodal forces by node by node lumping.

3.4. Analysis and post processing

Linear elastic analysis is prescribed. For the 2D problem plain strain conditions are prescribed. For liner 2D elements (triangular and quadrilateral) the full integration scheme has been prescribed (three and four integration points respectively) while for the 2D quadratic quadrilateral elements reduced integration is specified. For the 3D linear hexahedral elements reduced integration is specified (1 integration point) (although some participants eventually used full integration for the 3D problems). For the application of the structural stress calculations methods, presented in the following section, participants have been asked to use nodal values of the 1st principal stress (expect for the Dong's method). A unique value of the considered stress invariant should be obtained per node by averaging the nodal values from all the elements that share the node. However, the exact procedure for this averaging has not been defined.

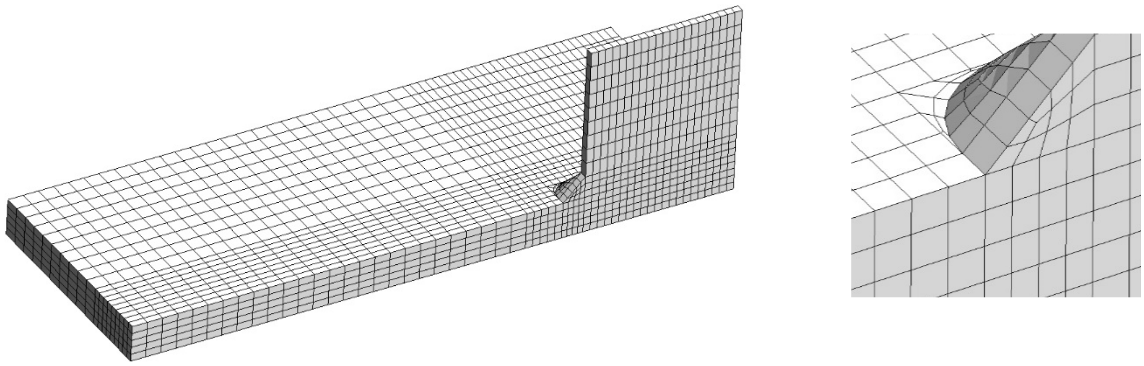


(a)

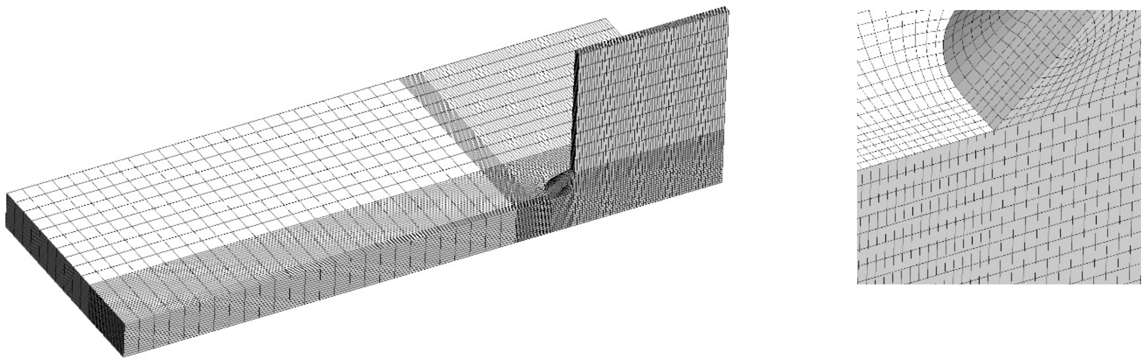


(b)

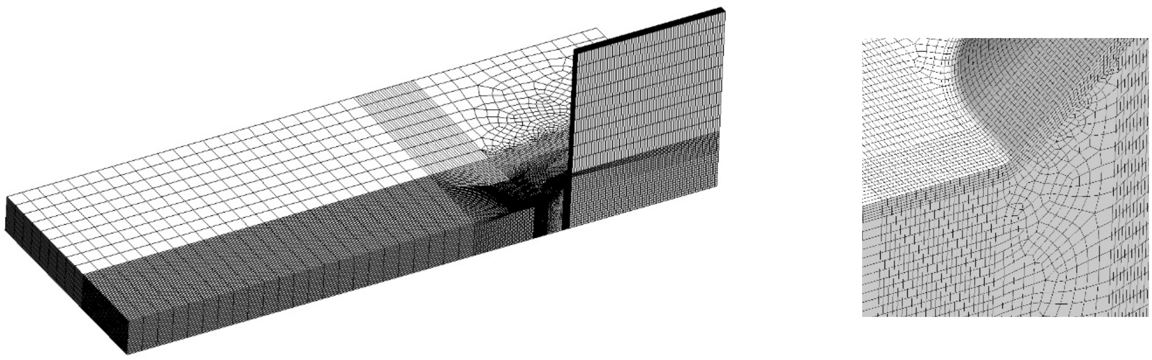
Fig. 2: Discretization of the 2D form.



Element length on surface = 2.4 mm. Element depth= 3 mm



Element length on surface = 0.4 mm. Element depth= 0.8 mm



Element length on surface = 0.13 mm. Element depth= 0.13 mm

Fig. 3: Discretization of the 3D form.

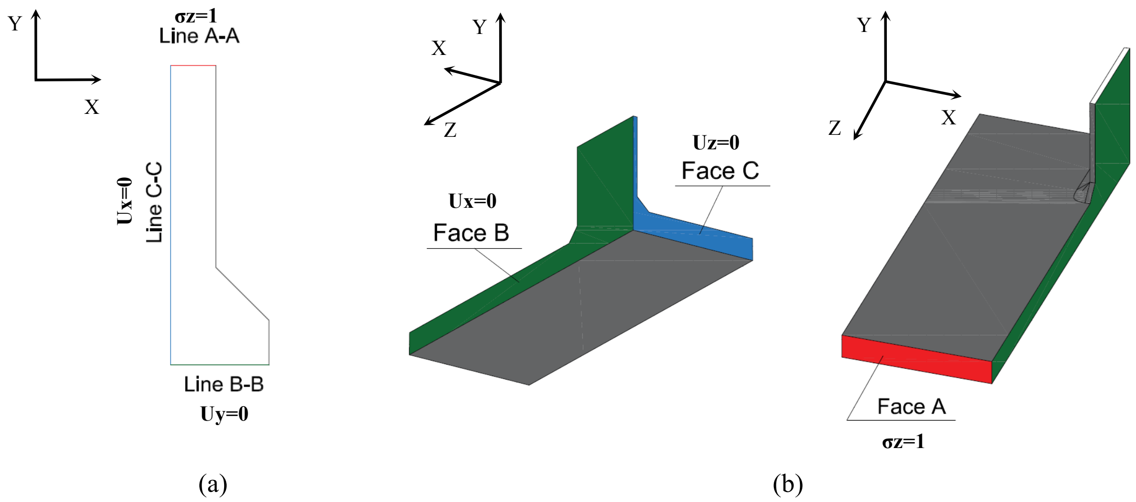


Fig. 4: Symmetry boundary conditions and loading (a) 2D problem, (b) 3D problem.

4. STRESS CONCENTRATION FACTOR CALCULATION METHODS

4.1. Hot spot method

Structural stress is calculated by linear extrapolation using the stress values obtained at a distance of 0.4 and 1.00 times the thickness of the element [6]. Other variations of the method also exist for application with coarser meshes and welds at plate edges. The structural stress calculated in the above way is supposed to include the effect of the element bending on the stress at the toe; but exclude the local effect due to the notch at the weld toe.

4.2. Modified Hot spot method

In the 2D case presented above it is clear that there is no bending in the loaded element unless eccentricity is explicitly modeled which is not the case here. Hence, the SCFs calculated with the above method are expected to be very close to the unit. Although this is absolutely normal it does not help the comparison between the various software which the objective of this study. For this raison it has been asked from the participants to calculate also a SCF according to a modified "hot spot" method. This method differs from the standard one only in the distances at which the stresses are evaluated. These distances are taken closer to the weld toe: at 0.2 and 0.5 times the thickness of the element. Of course, by using stresses closer to the point of stress concentration, the local effect of the notch begins to have a substantial contribution to the calculated SCF. It is not therefore easy to give a meaningful engineering interpretation of the SCF calculated by the above method. However, the obtain results may help to compare the various FE programs between them.

The appropriateness of the proposed meshes for the application of this method is investigated, for the 2D problem only, through a convergence analysis the results of which are summarized in Fig. 5. For this analysis meshes with uniform element size over the entire loaded plate have been used (additionally element size is the same in both directions). Linear triangular, linear quadrilateral, quadratic triangular and quadratic quadrilateral element are examined. In Fig. 5 it can be seen that in general acceptable convergence is achieved if the 0.2t evaluation distance coincides with the second (or higher rank) node from the weld toe (i.e. at least one quadratic or two linear elements between weld toe and 0.2t). An exception to that are the linear triangular element which require even smaller element size for achieving a good convergence.

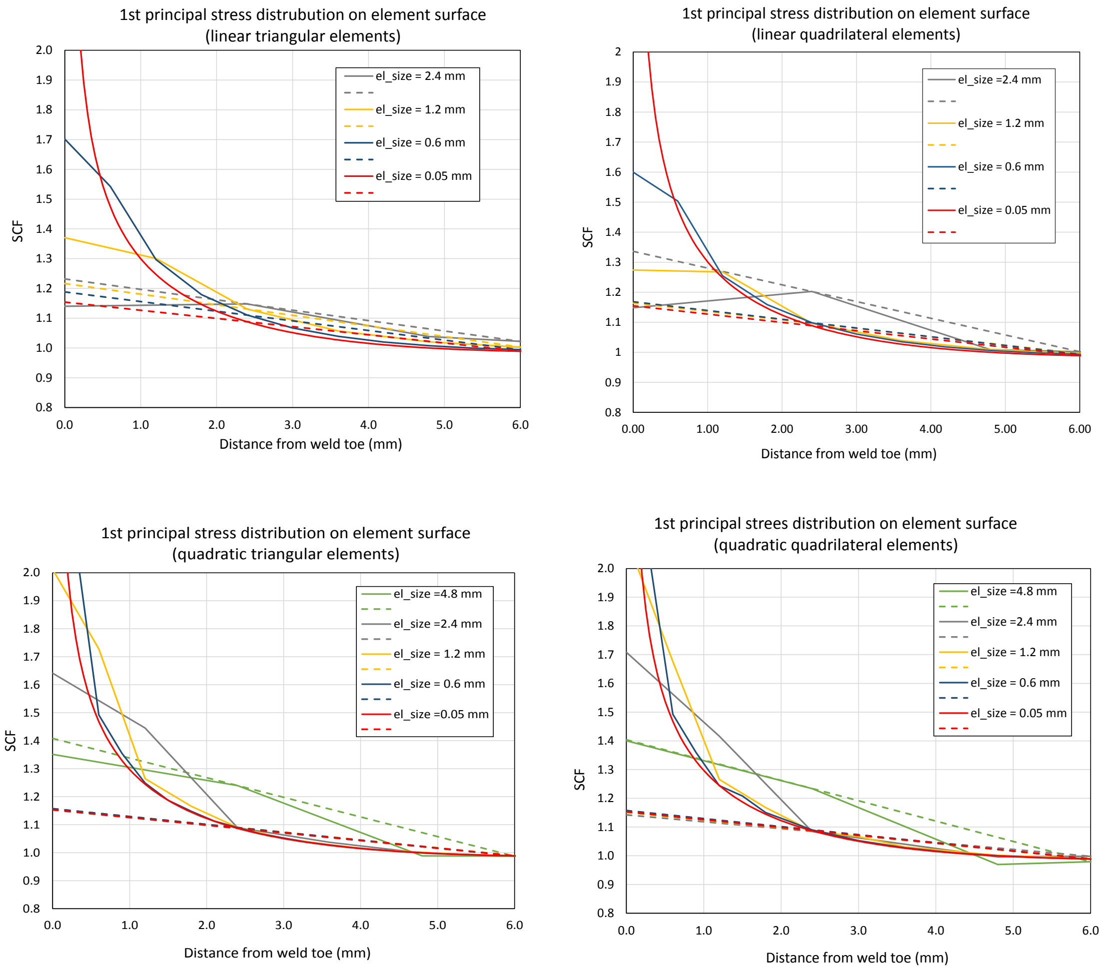


Fig. 5: Convergence analysis for the application of the modified "hot spot" method on the 2D problem. Dashed lines indicate the linear extrapolation to the weld root from values at 2.4 mm and 6 mm away from the toe.

4.3. Dong's method

Dong's method [9] consists in a through thickness linearization of normal stress distribution at a distance δ from the weld toe. The linearization is performed by considering the axial force bending moment equilibrium of the part of the element between the weld and the section at which stress are obtained. When the loaded plates are loaded in essentially axial load, so that the through thickens stress distribution is not monotonic, the linearization is applied up to a depth smaller than the total thickness. In particular, in symmetrically loaded element the linearization is applied to half of the thickness as shown in Fig. 6a. In that case the stresses on the symmetry line/plane should also be considered. The structural stress is calculated following the equations (with respect to Fig 6 a):

$$\sigma_{\mathrm{m}} = \frac{1}{t/2}\int_0^t \sigma_{\mathrm{x}} \mathrm{d}y$$

$$\sigma_{\mathrm{b}} = \frac{6}{(t/2)^2}\left(\int_0^{t/2} \sigma_{\mathrm{x}} y \mathrm{d}y + \delta \int_0^{t/2} \tau_{\mathrm{xy}} \mathrm{d}y + \int_0^{\delta} \sigma_{\mathrm{y}} x \mathrm{d}x - \sigma_{\mathrm{m}} \frac{(t/2)^2}{2}\right)$$

$$\sigma_{\mathrm{s}} = \sigma_{\mathrm{m}} + \sigma_{\mathrm{b}}$$

There is no generally accepted recommendation on what this distance should be, although in principle the result should be independent of this distance if the free body equilibrium is strictly imposed. In this study a distance δ=2.4 mm is adopted.

There is also considerable ambiguity on how Dong's method should be applied with results obtained from 3D FE models especially when there is a significant three dimensional variation of the stress field. For the 3D problem studied here, one approach is to apply the method as in the 2D case by considering only the nodal stress values on the YZ symmetry plane. In this case the structural stress for the studied 3D problem is given from the following equations (with respect to Fig 6 b):

$$\sigma_{\mathrm{m}} = \frac{1}{t}\int_0^t \sigma_{\mathrm{z}}\mathrm{d}y$$

$$\sigma_{\mathrm{b}} = \frac{6}{t^2}\left(\int_0^t \sigma_{\mathrm{z}} y\mathrm{d}y + \delta\int_0^t \tau_{\mathrm{yz}}\mathrm{d}y - \sigma_{\mathrm{m}}\frac{t^2}{2}\right)$$

$$\sigma_{\mathrm{s}} = \sigma_{\mathrm{m}} + \sigma_{\mathrm{b}}$$

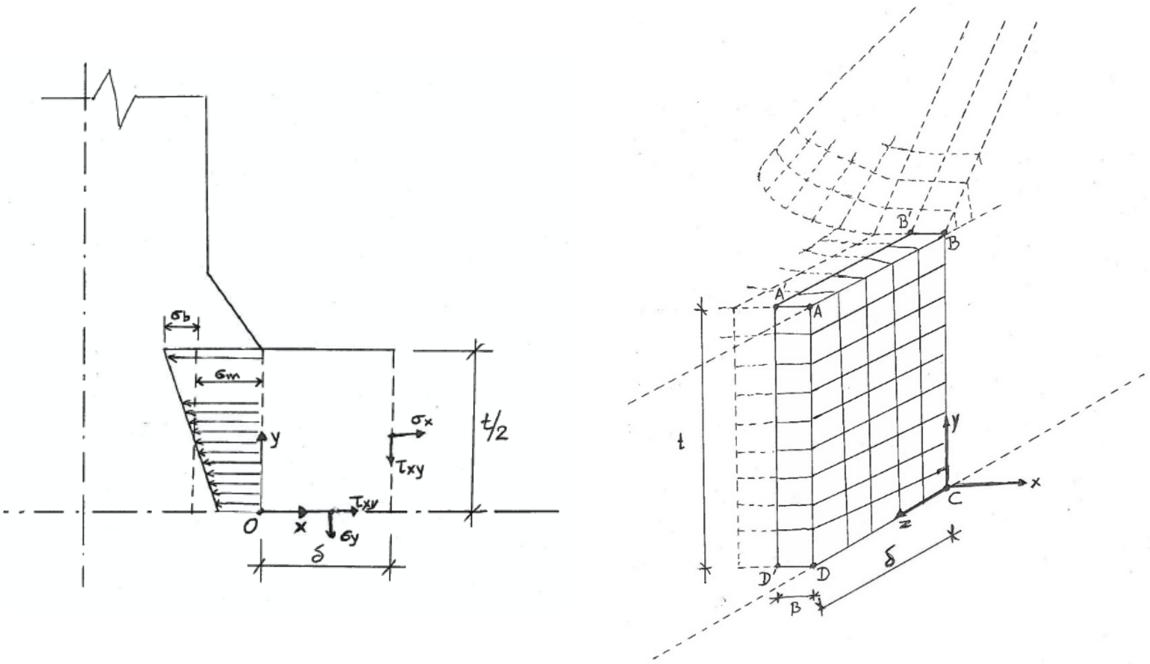
Note that this case the linearization is performed over the entire thickness of the element. This is the approach that has been used for the calculation of the round robin results.

However, a theoretically more consistent approach is to consider the equilibrium of a block of element adjacent to the symmetry plane as shown in Fig.6. Then the structural stress should be calculated as:

$$\sigma_{\mathrm{m}} = \frac{1}{\beta t}\left(\int_0^t y\int_0^\beta \sigma_{\mathrm{z}}\,\mathrm{d}x\,\mathrm{d}y + \int_0^t\int_0^\delta \tau'_{\mathrm{xz}}\mathrm{d}z\,\mathrm{d}y\right)$$

$$\sigma_{\mathrm{b}} = \frac{6}{\beta t^2}\left(\int_0^t y\int_0^\beta \sigma_{\mathrm{z}}\,\mathrm{d}x\,\mathrm{d}y + \delta\int_0^t\int_0^\beta \tau'_{\mathrm{yz}}\,\mathrm{d}x\mathrm{d}y + \int_0^t y\int_0^\delta \tau_{\mathrm{xz}}\mathrm{d}z\,\mathrm{d}y + \int_0^\delta z\int_0^t \tau_{\mathrm{xz}}\mathrm{d}y\,\mathrm{d}z - \sigma_{\mathrm{m}}\frac{\beta t^2}{2}\right)$$

$$\sigma_{\mathrm{s}} = \sigma_{\mathrm{m}} + \sigma_{\mathrm{b}}$$



(a) (b)

Fig 6: Dong's method application in the 2D cases.

4.4. Xiao and Yamada

Xiao and Yamada method is based on the observation in the absence of eccentricities and bending moments in the elements the local effect of the notch almost disappears at a distance of 1 mm below the surface, independently of the exact geometry of the weld toe. At the same time, at a structural level, this distance is sufficiently close to the surface so as to be a good approximation of the structural stress at the weld toe. The stress at that distance can therefore be considered as almost equal to the structural stress. In general, this method requires meshes which are much finer than those required for the hot spot method. A convergence analysis is preformed to evaluate the suitability of the proposed meshes for application with the Xiao and Yamada method. The results of this analysis are summarized in Fig. 7. These results show that, when linear element are used, acceptable convergence is achieved with just one element (of 1 mm size) between the surface and the stress evaluation point. On the other hand quadratic elements (and particularly the quadrilaterals) show a tendency for “overshooting” and require the use of at least two elements (of 0.5 mm size) between the surface and the stress evaluation point.

Hence, with the exception of the fine 2D mesh the meshes used in this study are not sufficiently fine for an accurate determination of the stress at 1 mm depth. The relevant results can only be used in order to compare the software between them.

The fact that the calculated structural stress (1.13 in this case) is not equal to one even though there is no moment in the elements is due to the fact that the initial assumption that the local effect completely disappears at 1 mm is not entirely true in this case. Hence depending on the general geometry of the examined joint the Xiao and Yamada structural stress may have a small influence from the local stress distribution at the notch.

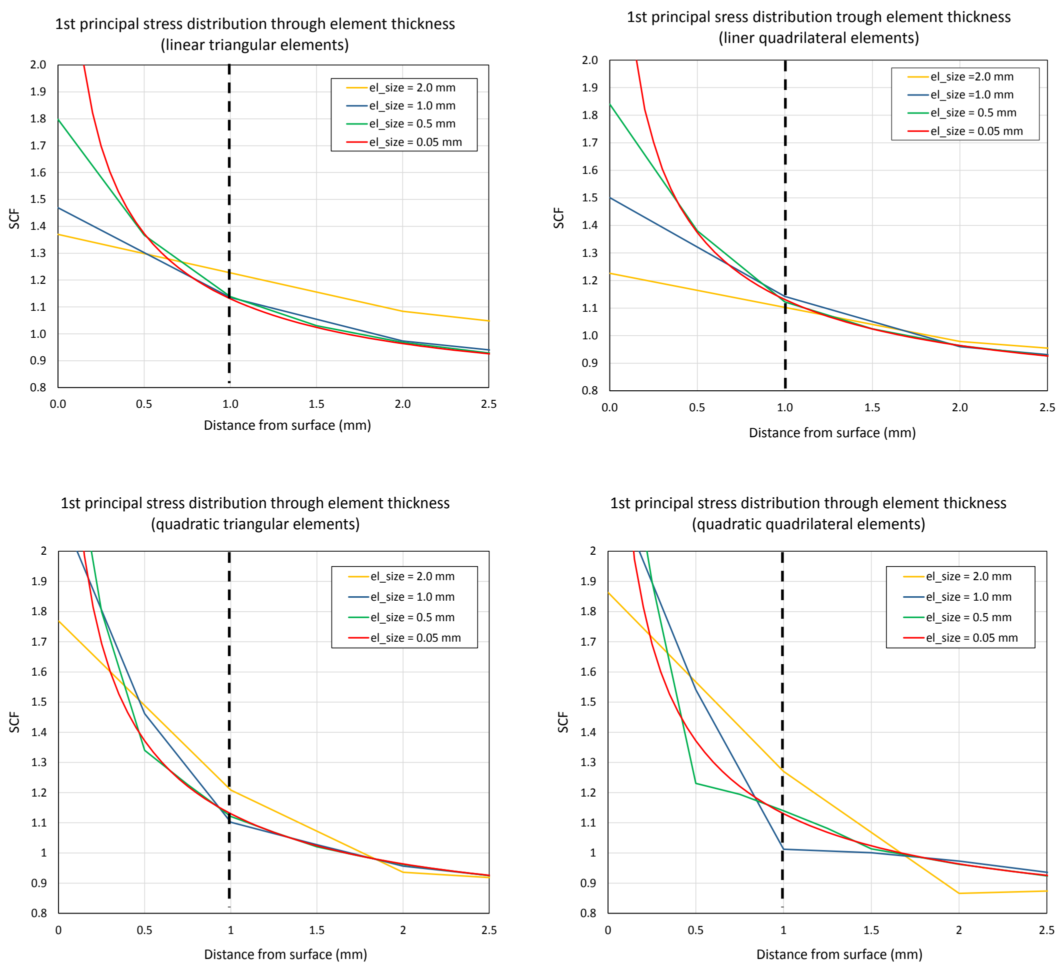


Fig. 7: Convergence analysis for the application of the Xiao and Yamada method on the 2D problem.

4.5. Haibach

The Haibach method is similar in concept to the Xiao and Yamada method and is based on the observation that at a distance of approximately 2 mm from the weld toe on the surface of the element the local stress effect due to the notch almost disappears while this point remains sufficiently close to the weld toe (at the structural level) so that the stress there is a good approximations of the structural stress at the weld toe.

A convergence analysis is also performed for the Haibach’s method and its results are summarized in Fig. 8. The analysis suggests that when linear element are used at least two elements (of 1 mm size) should exist between the stress evaluation point and the weld toe; when quadratic elements are used just one element (of 2 mm size) is sufficient for achieving acceptable convergence

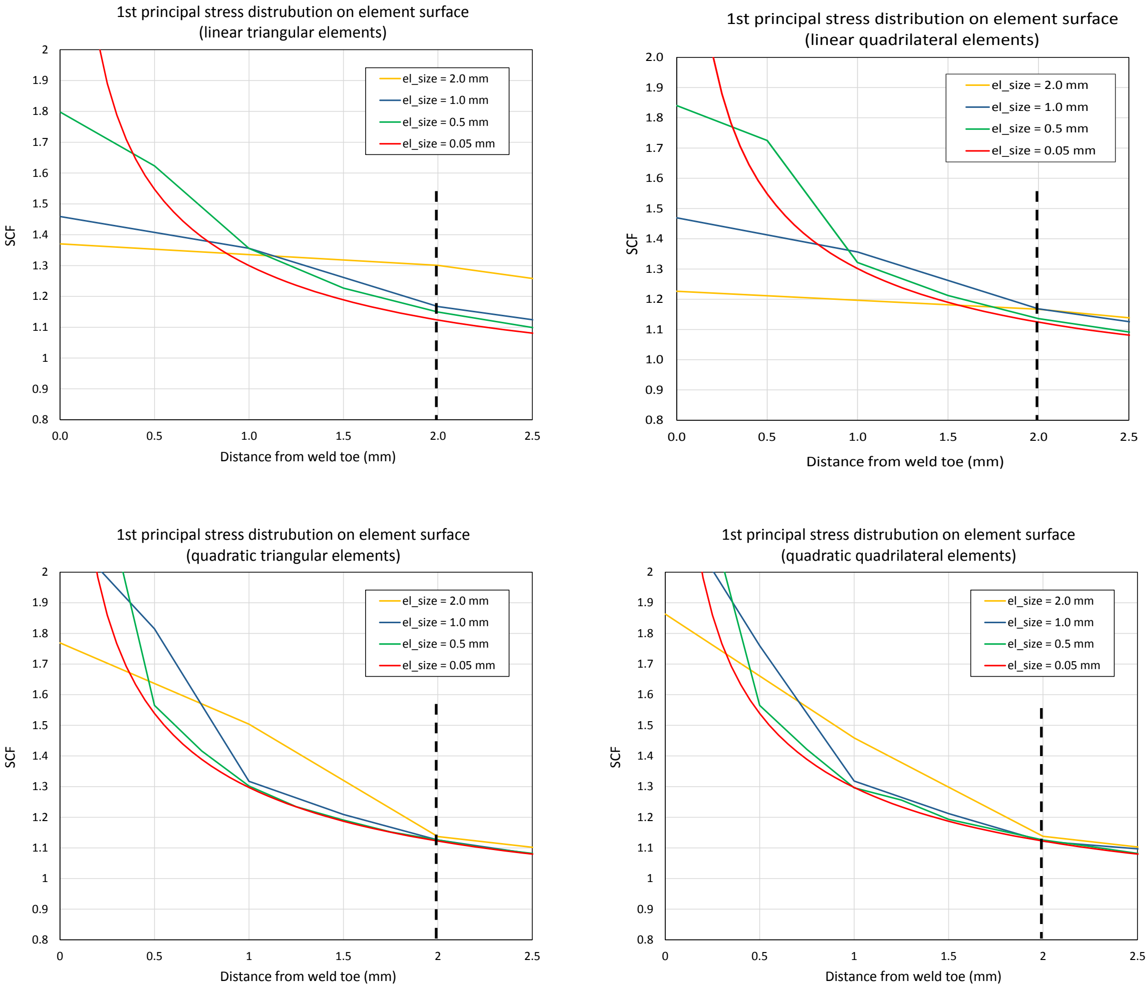


Fig. 8: Convergence analysis for the application of the Haibach's method on the 2D problem.

4.6. Critical distance

Contrary to the above methods the critical distance method seeks to calculate local stress which is representative of the stress state within an elementary material volume in which crack initiation takes place. This distance which in principle depends on the material microstructure is usually smaller than 1 mm. Hence the direct evaluation of the stress at the critical distance requires meshes even finer than those required by the Xiao and Yamada method. For this study the critical distance r_c proposed in [9] is used:

$$r_c = (\Delta K_{th}/\Delta\sigma_0)^2 \cdot (1/2\pi) = 0.296 \text{ mm}$$

Some of the participant reported also the values calculated for a critical distance $r_c = 0.01$ mm .

As it can be seen in Fig. 7 and 8, the "fine" meshes used in this study are not fine enough to be used for the local stress calculation at the critical distances. The above results presented here under the label critical distance should only be used for comparing the software between them and not as correct or meaningful values of the critical distance stress. For this reason, in a second stage of this round robin, additional "very fine" meshes (element size 0.05 mm) are proposed to the participants. As it can be seen in Fig. 9 these meshes are converged with respect to the stress at the critical distance $r_c = 0.296$ mm. At the time this revision is written, results for these additional analyses have be provided by only two of the participants.

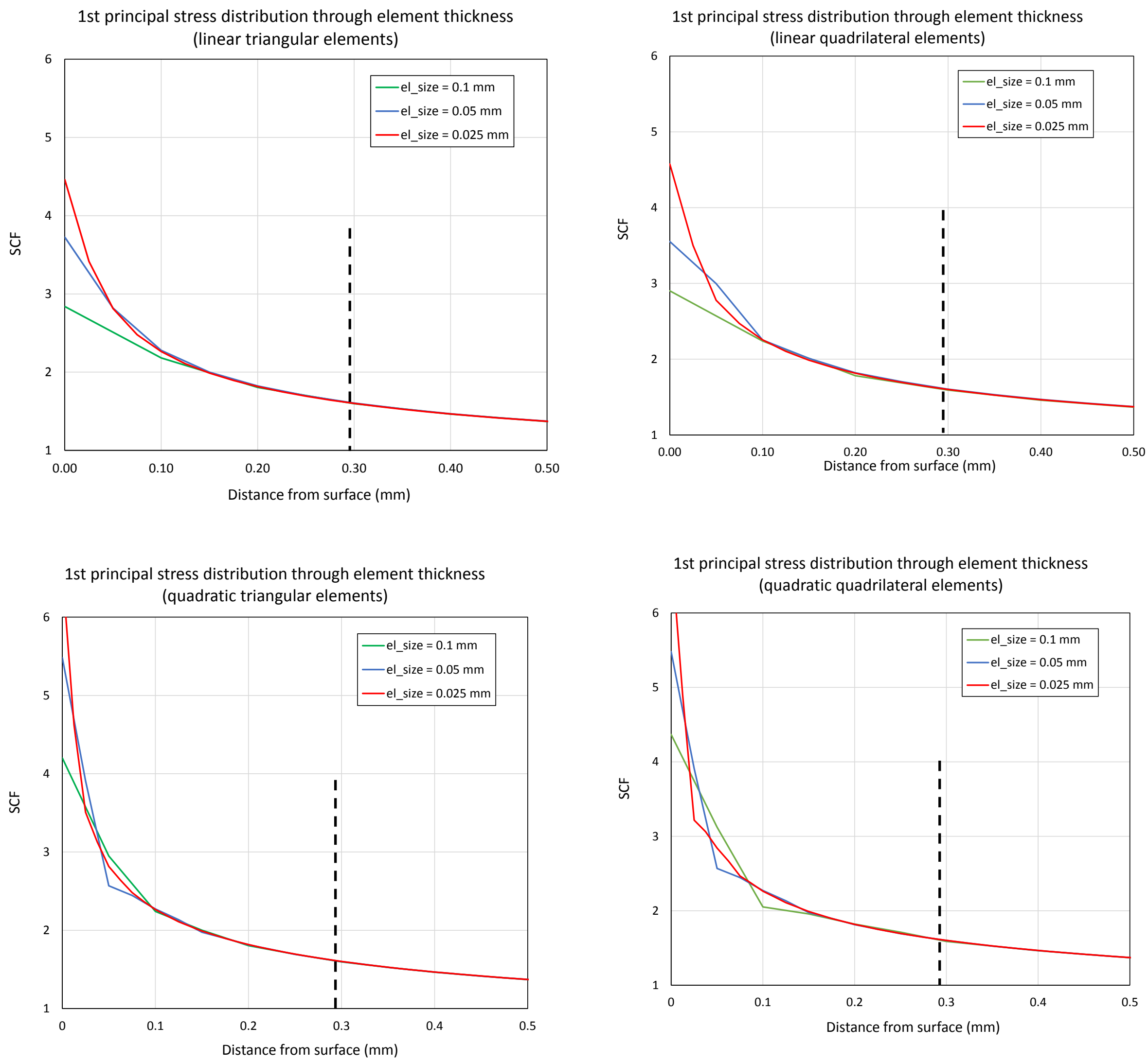


Fig. 9: Convergence analysis for the application of the critical distance method, with $r_c = 0.296$ mm, on the 2D problem.

4.7. Notch stress

The notch stress tries to evaluate the stress at the critical distance in an indirect way by assuming an equivalent filleted root instead of a sharp one. The radius of the fillet is properly selected so that the maximum stress on the surface of the fillet be equal to the stress at the critical distance. In this way coarser meshes can be used but the modeling is significantly more complicated since the geometry of the fillet needs to be also modeled and discretized. In this study a 1 mm radius is used for the fillets as proposed in [11].

Interestingly, the analysis shows that in the 3D problem the maximum stress does not occurs at the symmetry plane but where the weld quarter cone begins.

5. RESULTS

The results obtained from the participants are summarized in Tables 4 and 5 for the 2D and the 3D problems respectively. As it can be seen for the majority of the evaluated case there is a fairly good agreement between the participants with discrepancies being in general less than 10%. A number of larger discrepancies were explained by closely examining the applied modeling parameters and post processing – calculations procedure. The explanations are given in footnotes in the tables and are mainly related to the options that the various software give with respect to the stress-strain integration scheme. Poor agreement is observed in the evaluations of local stresses at the critical distance and many participant did not even reported values for these evaluation cases. This is not surprising since the proposed meshes (even the fines ones) are in fact too coarse for a stress evaluation so close to the weld toe. A small number of unexplained large discrepancies (marked in red in the table) still remain and require further investigation.

Table 2: Summary of the 2D plane strain models.

Mesh Geometry	Radius	Element Type	Integration Scheme	Methods	
Coarse	No	Quadratic Quadrilateral	Reduced	"Hot Spot"	use nodal values at 0.2 and 0.5 times the thickness on the surface – i.e. 2.4 mm and 6.0 mm
				Hot Spot	use nodal values at 0.4 and 1.00 times the thickness on the surface – i.e. 4.8 mm and 12.0 mm
				Xiao & Yamada	Interpolate using nodal values at weld toe and 1.5 mm below weld toe (below surface)
				Dong	Consider stress distribution at a distance 2.4 mm from weld toe. Integrate stresses through half of the thickness and use the stresses on the symmetry line
		Linear Triangular	Full	"Hot Spot"	use nodal values at 0.2 and 0.5 times the thickness on the surface – i.e. 2.4 mm and 6.0 mm
				Hot Spot	use nodal values at 0.4 and 1.00 times the thickness on the surface – i.e. 4.8 mm and 12.0 mm
				Xiao & Yamada	Interpolate using nodal values at 0.75 mm and 1.5 mm below weld toe (below surface)
				Dong	Consider stress distribution at a distance 2.4 mm from weld toe. Integrate stresses through half of the thickness and use the stresses on the symmetry line
		Quadratic Triangular	Full	"Hot Spot"	use nodal values at 0.2 and 0.5 times the thickness on the surface – i.e. 2.4 mm and 6.0 mm
				Hot Spot	use nodal values at 0.4 and 1.00 times the thickness on the surface – i.e. 4.8 mm and 12.0 mm
				Xiao & Yamada	Interpolate using nodal values at 0.75 mm and 1.5 mm below weld toe (below surface)
				Dong	Consider stress distribution at a distance 2.4 mm from weld toe. Integrate stresses through half of the thickness and use the stresses on the symmetry line
Fine	No	Quadratic Quadrilateral	Reduced	"Hot Spot"	use nodal values at 0.2 and 0.5 times the thickness on the surface – i.e. 2.4 mm and 6.0 mm
				Hot Spot	use nodal values at 0.4 and 1.00 times the thickness on the surface – i.e. 4.8 mm and 12 mm
				Xiao & Yamada	Interpolate using nodal values at 0.8 mm and 1.2 mm below weld toe (below surface)
				Dong	Consider stress distribution at a distance 2.4 mm from weld toe. Integrate stresses through half of the thickness and use the stresses on the symmetry line
				Critical distance	Interpolate using nodal values at weld toe and 0.2 mm below weld toe (below surface) (mid-node)
				Haibach	Use nodal value at 2.00 mm from weld toe on surface.
	Yes	Quadratic Quadrilateral	Reduced	Notch Stress	Maximum value on the notch fillet.

Table 3: Summary of the 3D models

Mesh Geometry	Radius	Element Type	Integration Scheme	Methods	
Coarse	No	Hexahedral	Reduced	Hot Spot	Use nodal values on YZ symmetry plane at 0.4 and 1.00 times the thickness on the surface – i.e. 4.8 mm and 12.0 mm
				Xiao & Yamada	Use nodal values on YZ symmetry plane. Interpolate using nodal values at weld toe and 2.4 mm below weld toe (below surface)

				Dong	Use only nodal values on YZ symmetry plane Consider stress distribution at a distance 2.4 mm from weld toe. Integrate stresses through the entire thickness of the plate.
Fine	No	Hexahedral	Reduced	Hot Spot	Use nodal values on YZ symmetry plane at 0.4 and 1.00 times the thickness on the surface – i.e. 4.8 mm and 12.0 mm.
				Xiao & Yamada	Use nodal values on YZ symmetry plane. Interpolate using nodal values at 0.8 and 1.6 mm below weld toe (below surface).
				Dong	Use only nodal values on YZ symmetry plane. Consider stress distribution at a distance 2.4 mm from weld toe. Integrate stresses through the entire thickness of the plate.
				Critical distance $r_c = 0.296$ mm	Use only nodal values on YZ symmetry plane. Interpolate using nodal values at weld toe and 0.8 mm below weld toe (below surface).
				Critical distance $r_c = 0.100$ mm	Use only nodal values on YZ symmetry plane. Interpolate using nodal values at weld toe and 0.8 mm below weld toe (below surface).
				Haibach	Use only nodal values on YZ symmetry plane. Interpolate using nodal values at 0.8 mm and 1.6 mm from weld toe on surface.
	Yes	Hexahedral	Reduced	Notch Stress	Maximum value on the notch fillet (not necessarily on the YZ symmetry plane).

6. CONCLUSIONS

A non-negligible influence of the software on the evaluated structural/local stress has been revealed. It seems that if all the basic modeling parameters are thoroughly specified the discrepancies can be limited to approximately 10%.

The hot spot method and the "hot spot" method give the most consistent results between various software among all the examined methods. This is reasonable since this method is the less affected by the local stress concentration due to the weld toe and the meshes were well adapted for the application of this method. On the other hand the poorest agreement is observed for the critical distance method (with the relatively coarse for this method meshes that were used) for which the influence of the local stress consecration is maximal. It seems that the quality of meshing influences not only the accuracy but also the consistency of the results.

The experience of this round robin has shown that the results may considerably depend on the various analysis options. Even if all the important modeling and analysis options are specified in detail, which is a rather complicated task in itself, increased attention is required by the analyst in order to fully comply with these modeling and calculations specifications. Important variation in the results may be due (among other reasons) to different practices in relation to: the integration scheme, elements that are considered in the nodal averaging, the use of the results at mid-nodes.

Eventually, The two problems studied here in this round robin could be proposed as benchmark examples for the validation of FE modeling procedures aiming at structural stress evaluation by engineers how are not familiar with the structural stress methods.

ACKNOWLEDGEMENTS

This round robin would not, of course, have been possible without the interest and the efforts of all the participants (listed in Table 1). Their contribution to this work has been extremely important and is highly acknowledged. The authors we would also like to thank all the participants for the fruitful discussions and for sharing their point of view and their comments on the topic and the results of this round robin.

The authors we would also like to thank our colleague Mr. Albano de Castro e Sousa for the extremely important work that he did for the launching of this round robin, including the preparation of meshes and the report sheets, for the admiration of the first phase of the round robin and for his contribution to the results.

Table 4: Results for the 2D cases.

				participant	1		2			3	4	5	6
				software	a	b	a	b	c				
Mesh Geometry	Radius	Element Type	Integration Scheme	Methods									
Coarse	No	Quadratic Quadrilateral	Reduced	“Hot Spot”	1.16	1.16	1.16	1.16	1.33(1)	1.16	1.16	1.18	1.16
				Hot Spot	1.00	1.01	1.00	1.00	1.00	1.00	1.01	1.02	1.00
				Xiao & Yamada	1.22	1.22	1.22	1.22	1.10(1)	1.22	1.26(3)	1.21	1.22
				Dong	1.30	1.30	1.27	1.27	1.28	1.30	1.34	1.27	
		Linear Triangular	Full	“Hot Spot”	1.25	1.25	1.25	1.20	1.59(2)	1.26	1.25	1.25	1.25
				Hot Spot		1.06	1.06	1.05	1.14(2)	1.07	1.06	1.06	1.06
				Xiao & Yamada	1.14	1.13	1.14	1.14	1.08(2)	1.13	1.13	1.14	1.12
				Dong	1.34	1.34	1.28	1.28	1.29	1.26	1.33	1.29	
		Quadratic Triangular	Full	“Hot Spot”	1.21	1.21	1.21	1.21	1.17(2)	1.21	1.21	1.22	1.20
				Hot Spot		1.01	1.01	1.01	1.00	1.00	1.01	1.04	1.01
				Xiao & Yamada	1.21	1.22	1.15	1.22	1.13	1.21	1.26	1.21	1.21
				Dong	1.28	1.31	1.29	1.29	1.30	1.32	1.35	1.28	
Fine	No	Quadratic Quadrilateral	Reduced	“Hot Spot”	1.15	1.15	1.15	1.15	1.15	1.16	1.15	1.17	1.15
				Hot Spot	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.01	1.00
				Xiao & Yamada	1.14	1.15(4)	1.33	1.15	1.13	1.15	1.14	1.15	1.15
				Dong	1.34	1.34	1.26	1.26	1.27	1.30	1.34	1.26	
				cr. dist. $r_c = 0.296$ mm		1.75	2.01	1.68	2.01		1.75	1.75	
				cr. dist. $r_c = 0.100$ mm		2.52	2.65	2.50	3.77(1)		2.52		
				Haibach	1.12	1.12	1.19	1.18	1.17		1.12	1.12	
	Yes	Quadratic Quadrilateral	Reduced	Notch Stress	3.05	2.58	2.58	2.61	2.59	2.58	2.58	2.58	2.58
Very fine	No	Quadratic Quadrilateral	Reduced	“Hot Spot”		1.15					1.15		
				Xiao & Yamada		1.13					1.13		
				cr. dist. $r_c = 0.296$ mm		1.61					1.60		
				Haibach		1.12					1.12		
		Linear Triangular	Full	“Hot Spot”		1.15					1.16		
				Xiao & Yamada		1.13					1.13		
		Quadratic Triangular	Full	“Hot Spot”		1.15					1.15		
				Xiao & Yamada		1.13					1.13		

(1) The discrepancy from the rest of the results is most probably due to: (a) the fact that this software (Comsol 4.4) does not offer the possibility of reduced integration; (b) the fact that for the post processing the software applies a an accurate derivative recovery method by Z. Zhang [] instead of simple averages.

(2) The discrepancy from the rest of the results is most probably due to the application of the accurate derivative recovery method as above.

(3) Value obtained by interpolation between the corner node at weld toe and the first element corner node at a distance of 1.5 mm (instead of interpolating between the first mid node at 0.75 mm and the first corner node at 1.5 mm).

(4) When the value at the mid node at a distance of 1 mm is directly taken instead of interpolating between corner nodes a value of 1.15 is obtained.

Table 5: Results for the 3D cases.

					1		2			3	4	5	6
					a	b	a	b	c				
Mesh Geometry	Radius	Element Type	Integration Scheme	Methods									
Coarse	No	Brick	Reduced	Hot Spot	1.33	1.33	1.33	1.36	1.32(1)	1.42(2)	1.35	1.33	1.48
				Xiao & Yamada	1.32	1.28	1.32	1.31	1.69(1)	1.52(2)	1.31	1.32	1.22
				Dong	1.31	1.40	1.42	1.42	1.58(1)	1.42(2)	1.45	1.44	
Fine	No	Brick	Reduced	Hot Spot	1.36	1.36	1.37	1.37	1.38(1)	1.37(2)	1.39	1.36	1.38
				Xiao & Yamada	1.57	1.57	1.57	1.43	1.46(1)	1.31(2)	1.60(4)	1.57	1.41
				Dong	1.41	1.42	1.42	1.43	1.43(1)	1.42(2)	1.45	1.38	
				cr. dist. r_c = 0.296 mm		1.89	2.16	3.04	3.04(1)	2.30(2)	1.98(4)	1.88	
				cr. dist. r_c = 0.100 mm		1.93	1.99	2.44	3.21(1)	-		1.45	
				Haibach	1.47	1.47	1.53	1.64	1.72(1)	1.50(2)	1.49(4)		
	Yes	Brick	Reduced	Notch Stress	2.89	2.89(3)	2.90	3.29	3.18(1)	3.33(2)	2.97	2.89	3.36(2)

(1) Full integration and accurate derivative recovery method have been applied.

(2) Full integration has been applied.

(3) If full integration is applied instead of reduced a value of 3.31 is obtained.

(4) Only elements in front of the weld considered (elements below the weld do not contribute to the nodal values).

REFERENCES

[1] Doerck O., Fricke W., & Weissenborn C. (2003). Comparison of Different Calculation methods for Structural Stresses at Welded Joints. Int. J. of Fatigue, vol. 25.

[2] Poutiainen, I., Tanskanen, P., and Marquis, G. (2004). Finite element methods for structural hot spot stress determination—a comparison of procedure, International Journal of Fatigue, vol. 26 ,pp. 1147–1157

[3] Fricke, W, and Kahl, A. (2005). Comparison of different structural stress approaches for fatigue assessment of welded ship structures, Marine Structures, vol 18, pp473-488.

[4] Fricke, W. (2007). Round-Robin Study on Stress Analysis for the Effective Notch Stress Approach, Welding in the World , Vol. 51, Issue 3-4 (March), pp 68-79

[5] Mettänen, H., Björk, T., and Nykänen, T. (2013). Comparison of different methods for fatigue assessment of vibrating welded structures, IIW-Doc. XIII-2474-13.

[6] Hobahair

[7] Xiao, Z.-G., Yamada, K. (2004). A method of determining geometric stress for fatigue strength evaluation of steel welded joints, Int. J. of Fatigue, vol. 26, pp. 1277–1293

[8] FKM Guideline, Fracture mechanics proof of strength for engineering components, third revised edition, English version, Eds. Forschungskuratorium Maschinenbau, VDMA Verlag GmbH, Frankfurt/Main, 2009.

[9] Dong P. (2001). A Structural Stress Definition and Numerical Implementation for Fatigue Analysis of Welded Joints. Int. J. of Fatigue, vol. 23. 2001.

[10] Taylor, D., and G. Wang. The Validation of Some Methods of Notch Fatigue Analysis. *Fatigue and Fracture of Engineering Materials and Structures* 23, no. 5 (2000): 387–94.

[11] Baumgartner, J., and Bruder, T. (2010). An efficient meshing approach for the calculation of notch stresses, IIW-Doc. No. XIII-2313-10

[12] Niemi E., Fricke W. & Maddox S.J. (2006). Fatigue Analysis of Welded Components: Designer's guide to the structural hot-spot stress approach. doc. IIW-1430-00, Woodhead Publishing Ltd.

[13] Naga, A., and Zhang, Z. (2005). The Polynomial-Preserving Recovery for Higher Order Finite Element Methods in 2D and 3D, Discrete and Continuous Dynamical Systems—Series B, vol. 5, pp. 769–798, 2005.