EFFICIENT AND CONFORMING-TO-STANDARD FATIGUE ASSESSMENT OF WELDED STRUCTURES USING UNSTRUCTURED CONTINUUM ELEMENT MESHES

Dr. N. Friedl, Dr. W. Vonach, P. Löffler (CAE Simulation & Solutions GmbH, Austria)

THEME

Durability, Fatigue & Fracture

SUMMARY

Regarding FEM for welded structures, shell elements are mostly used to capture the structural behavior very efficiently. Depending on the choice of the element order, mesh size, etc. local nominal or even hot spot stresses can be calculated. These values can be assessed directly according to typical standards, i.e. Eurocode, DVS, FKM-Guideline etc.

In practical applications there can be strong arguments against the shell element approach. The effort to generate an appropriate mid surface shell model often is significant. Automated mid surface generation is common standard, but the correct joining of the parts (ie the 'welding' of the model) together with the necessity for a good mesh quality and size in these most critical welding areas still cannot be automated in a satisfactory manner. For complex structures costly manual work is always necessary.

Thus it is inviting to mesh the CAD volumes directly and work with continuum elements. The corresponding meshes are available in much shorter time. But when it comes to the fatigue stress assessment, the gain in efficiency is usually lost. Mostly, one ends up with stress results that are difficult to categorize regarding fatigue, since they are a mixture of nominal and partly resolved structural hot spot stresses or even singularities. The permissible values from fatigue standards are thus not directly applicable to these FEM stress results.

In our contribution we present a method, which allows an efficient fatigue assessment of welded structures, based on deriving relevant stress quantities from the displacement results of continuum models. This method has been implemented in the commercial software LIMIT for typical standards.

Using structural hot spot stresses simplifies the choice of FAT classes since the structural behavior is fully resolved by the Finite Element model. Additionally the novel approach loosens the strong restrictions on element type and size usually involved with structural hot spot stress approaches.

Test cases show a good agreement with results obtained by procedures from literature.

The new method enables significant time and cost saving in the simulation cycle of fatigue assessment of welded structures by avoiding the common approach of mid-surface shell modeling of welded structures.

KEYWORDS

Fatigue, Postprocessing, Welding, Structural Hot Spot Stress, Nominal Stress.

1: INTRODUCTION

In numerous fields of mechanical engineering sufficient fatigue strength of structures is an important issue. Almost every industrial sector has its own specific standards and guidelines in order to calculate margins of safety against fatigue failure. In particular welded structures are susceptible to fatigue, since this type of joining technology often goes hand in hand with sharp notches and low fatigue strength values.

1.1 State of the art fatigue analysis of welded structures

Welded structures are usually made of plate material with large dimensions compared to their thickness. For the Finite Element approach shell elements can be used to capture the structural behaviour very efficiently. The shell elements are placed at the mid surfaces of the plates. Depending on the choice of the element order, the number of integration points and the mesh size the results can be interpreted as local nominal or even hot spot stresses. These values can be assessed according to typical standards, i.e. Eurocode 3, IIW, DVS1612, FKM-Guideline and many more. The permissible stress ranges for different weld types can directly be compared with the calculated FEM stress values leading to margins of safety at each assessment position.

1.2 Analyzing the workflow

In practical applications there can be strong arguments against the shell element approach discussed above. The geometry of the structure is usually defined as a full 3D CAD model. The effort to generate an appropriate mid surface model is significant and costly. Figure 1, left upper corner, shows a typical CAD model of a welded structure. Automated mid surface generation is the common and widespread preprocessing tool used here for the first shell modeling step. But the joining of the parts (i.e. the ‘welding’ of the model) together with the necessity for a good mesh quality and mesh size in these most critical welding areas still cannot be automated in a satisfactory manner.

For complex structures manual work always is necessary, as well as rather expensive specialized preprocessors. Therefore the preprocessing often is the most expensive part of the simulation cycle. Furthermore some regions of the structure can violate the assumptions of shell theory when their thickness is large as compared to other dimensions. Typical examples are castings at load introductions or strong geometrical discontinuities. Transforming these structures to shell models might affect the overall behaviour since the local stiffness and load carrying mechanisms are influenced.

For the reasons mentioned above many engineers prefer to work with continuum elements and mesh volume geometries directly, mostly after removing some local features like holes or fillets. That way meshes for the Finite Element analysis are available in much shorter time. With most preprocessors the volume mesh is produced very fast by using automatic cleanup tools in combination with the robust tetrahedral meshing algorithm. The automatic contact detection tool is most helpful to define the necessary TIE constraints for joining (welding) the parts together with a few mouse clicks. Figure 1 gives an example for a tetrahedron mesh, generated with global mesh seed in a very short time.

As compared to shell models, such volume models are of course computationally more expensive. In our experience this drawback is continually diminishing. Using modern but moderately priced hardware in combination with the continually improving solvers, such models can be handled with acceptable computing time, even if a considerable number of loadcases and some non-linearities (e.g. contact in bolted flanges,…) are present.

But when it comes to the fatigue stress assessment the gain in efficiency is lost. Depending on the mesh parameters (order of elements, size, resolved geometric features) one ends up with stress results that are very difficult to categorize and assess. Nominal stresses or hot spot stresses are not directly available after the Finite Element analysis. The permissible values from fatigue standards are thus not applicable.

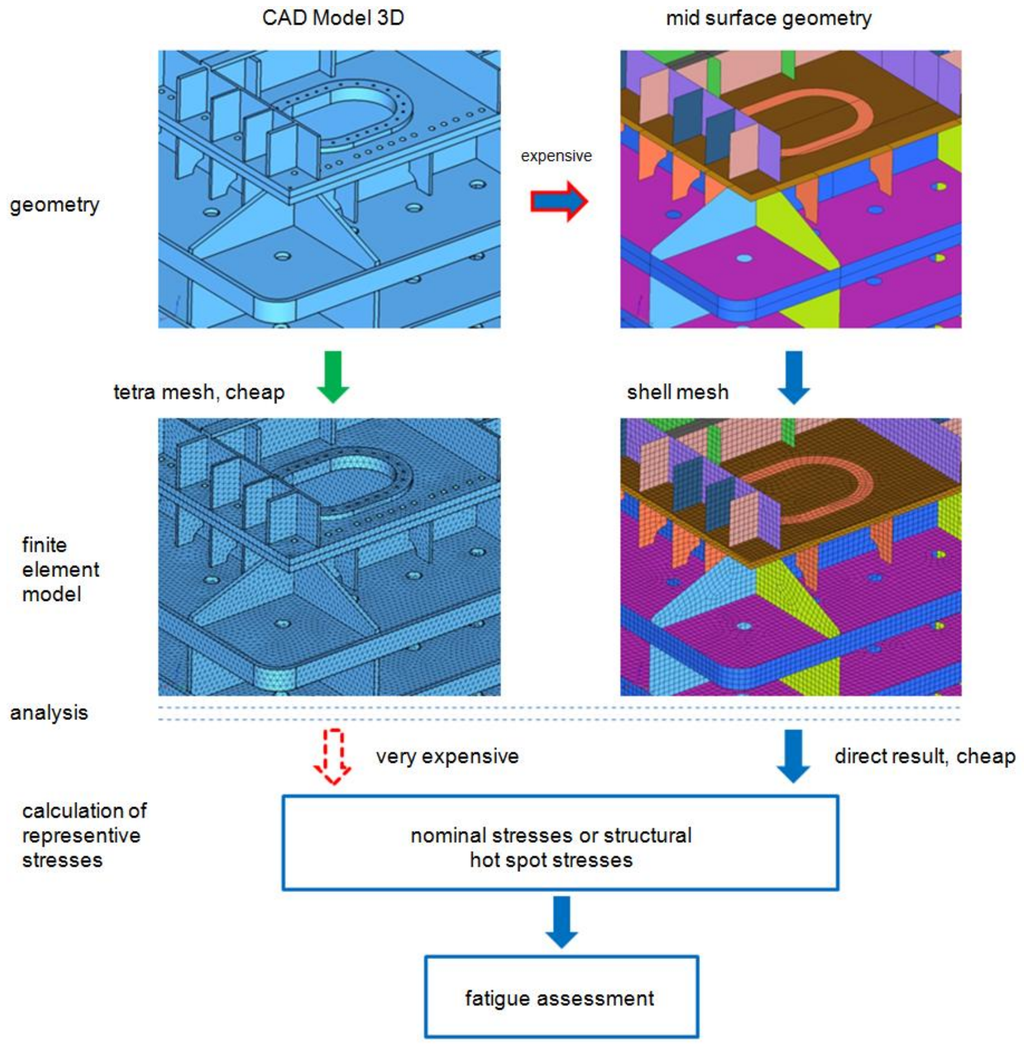


Figure 1: Work flow fatigue assessment via shell or tetra mesh model

In order to perform a fatigue assessment according to a typical standard one has to add another post processing step in deriving the relevant stress quantities (e.g. nominal stress) from the simulation results of the solid elements. Today, this step is time expensive semi automated work. This assessment is therefore usually limited to a small number of assessment positions which may not capture all fatigue hot spots in the structure correctly.

Results of shell elements, on the other hand, can be used directly for the fatigue assessment, since their results (appropriate meshing provided) are a good representation of local nominal stresses or even structural hot spot stresses.

1.3 Objective of the project

While delivering fatigue assessment services and tools to our customers, we developed different strategies to raise our efficiency. In this project we are aiming at the following:

- Minimal preprocessing cost using 'Tet and tie meshes':
    - tetrahedron meshes, mesh size above wall thickness
    - no special treatment of welds: either merging of CAD bodies or TIE constraints to connect the welded bodies
- Fatigue assessment according to standards at sufficient accuracy

2: FATIGUE ASSESSMENT FOR WELDS IN SOLID MODELS

2.1 Nominal and structural stresses

For fatigue assessment of welded structures two stress quantities are mainly used in practical applications.

(a) Nominal stresses,

(b) Structural hot spot stresses.

Figure 2 defines the different stresses according to the recommendations of the International Institute of Welding (IIW) (Hobbacher, 2007).

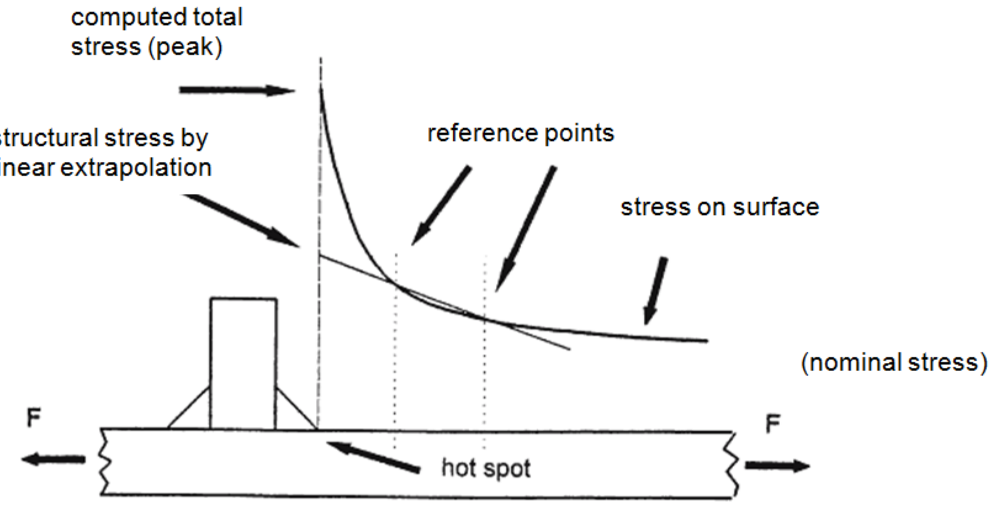


Figure 2: Stresses at welded joint (Hobbacher, 2007)

For each stress type, i.e. nominal or structural, FAT classes can be found, defining the fatigue resistance for a special welding detail. Figure 3 shows permissible nominal stresses for a reinforcement plate. In this case nominal stresses are calculated only with the cross section of the hollow section.

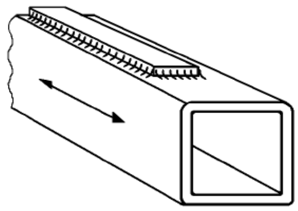
721		End of reinforcement plate on rectangular hollow section. wall thickness: t < 25 mm	50

Figure 3: Fatigue resistance of doubling plate, nominal stress, (Hobbacher, 2007)

Usually one would add the reinforcement plate in the Finite Element model of the beam. In this case the stresses obtained near the reinforcement plate are no longer nominal stresses because they include the influence of the reinforcement (stress concentration, induced bending, etc…). In order to assess this detail structural hot spot stresses are used. Figure 2 shows the method, details on the position of the reference points are given in (Hobbacher, 2007). At the two different reference positions stress values are determined. Via linear extrapolation the structural stress can be found for the weld toe (hot spot). This value can be compared to the fatigue resistance found for structural stresses (Figure 4). As can be seen, the permissible structural stress (FAT 100MPa) is much higher than the nominal one (FAT 50MPa).

5		Cover plate ends and similar joints	As welded	100

Figure 4: Fatigue resistance of doubling plate, structural hot spot stress, (Hobbacher, 2007)

2.2 Structural hot spot stresses from solids

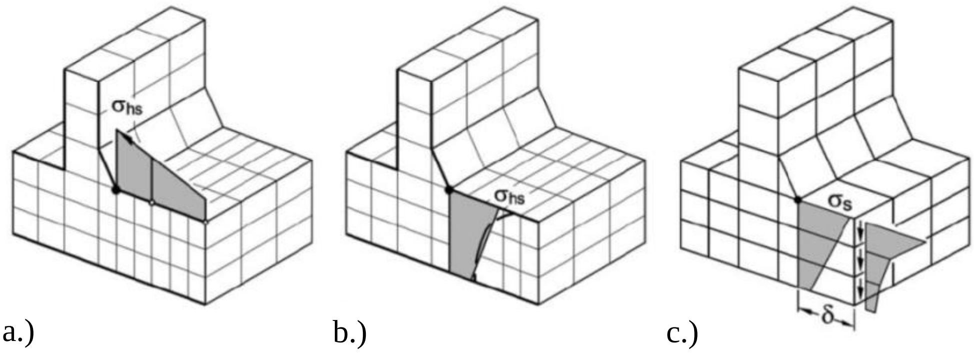


Figure 5: Different approaches for structural hot spot stresses (Fricke, 2005)

In the literature there are different procedures for the computation of structural hot spot stresses. Three typical methods are discussed in (Fricke, 2005) and shown in Figure 5. The method a.) is an example for the linear extrapolation along the surface according to IIW (Hobbacher, 2007). Method b.) shows the calculation of structural stresses directly at the weld toe. In both cases the nonlinear stress distribution through the plate thickness has to be linearised while maintaining the section forces and moments to gain the appropriate assessment stress value. Method c.) was introduced in (Dong, 2006). In this case the section forces and moments are calculated at a distance δ from the toe, away from high local stress peaks. The out of plane section force can be interpreted as first derivative of the local bending moment. This way the bending moment at the toe can be derived and the assessment stress value can be obtained. Methods b.) and c.) can also be formulated using nodal forces instead of element stresses.

All three cases work best with hexahedron element meshes, aligned with the local weld details and fairly small element size. Unstructured tetrahedron meshes could be used in combination with a.) but extensive numerical studies at CAE Simulation & Solutions (Löffler, 2012) have shown, that the extrapolation of stress data gained from tetrahedral meshes is not reliable (see results in Figure 10: i and j).

Figure 6 shows a doubling plate with a coarse mesh of the merged parts. a.) shows non averaged stresses. The results are not continuous over element boundaries. Usually averaged stresses (b) are used for post processing issues. However, by averaging the peak stresses near the weld toe are reduced significantly (red arrow, Figure 6).

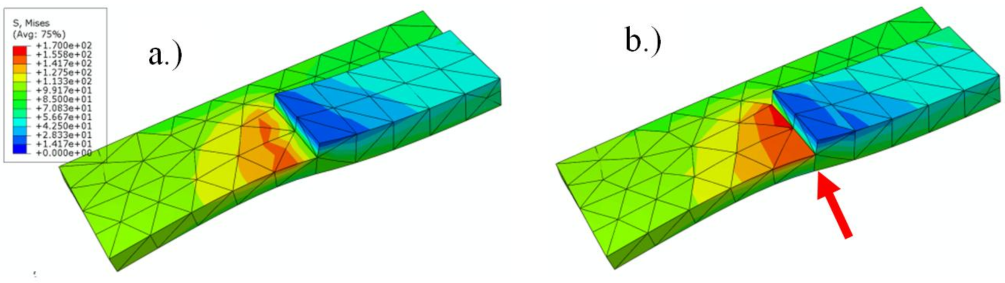


Figure 6: Averaged and non averaged results, doubling plate, coarse mesh

The IIW extrapolation scheme (Figure 5, case a) based on non averaged FEM stress results works well if the first row of tetrahedron elements lies within 0.3 times the shell thickness (which is a drastically finer mesh as depicted in Figure 6). However, this constraint is too limiting for practical use.

3: EMBEDDED SHELL SENSOR

3.1 Sensor element approach

In order to overcome previously mentioned problems, we decided to use a novel approach.

In standard Finite Element procedures, the primary variables solved for in a structural analysis, are displacements at the nodes. With the help of interpolation functions the displacement field within the element is defined. The displacement fields are continuous across element borders. In our method, we calculate the structural stresses from the displacement fields of the solid structure. In the post processing phase, we introduce sensor shell elements, embedded within the solid structure. Two sensors can be seen in Figure 7. The sensors are placed along the mid surface of the plate. Each assessment point has its own sensor.

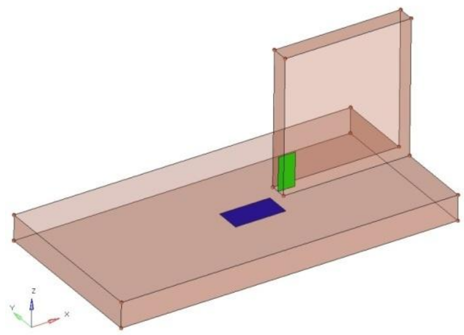


Figure 7: Position of the sensor shell element

This method is independent of the underlying solid mesh and the stress extraction points can be positioned arbitrarily (e.g. according to IIW recommendation). The sensor element is deformed by the displacement field of the surrounding elements and delivers the structural hot spot stress or the local shell section force.

Advantages of this procedure:

(a) The only constraint to the method is a sufficiently resolved displacement field,

(b) The sensor is independent of underlying mesh and element type,

(c) The method also works near TIE constraints,

(d) The sensor size can also define the region of averaging stresses. This way even approximations of nominal stresses or stresses comparable to shell results can be derived

3.2 Placing the sensor elements

The real challenge in the procedure is placing the sensor elements in the model. CAE Simulation & Solutions has developed a Postprocessing tool which supports an automated and quick way to introduce the sensor elements even in large structures. Figure 8 shows a typical welding situation namely a T-joint. Red circles mark potential areas of crack initiation. The T-joint can be separated into three branches, each of them carrying moments and forces to the position of the fillet weld. Embedding sensors in every branch give the section forces and moments needed to calculate structural stresses in the possible crack locations or even net stresses in the fillet. This way all critical positions can be analyzed also considering the weld thickness.

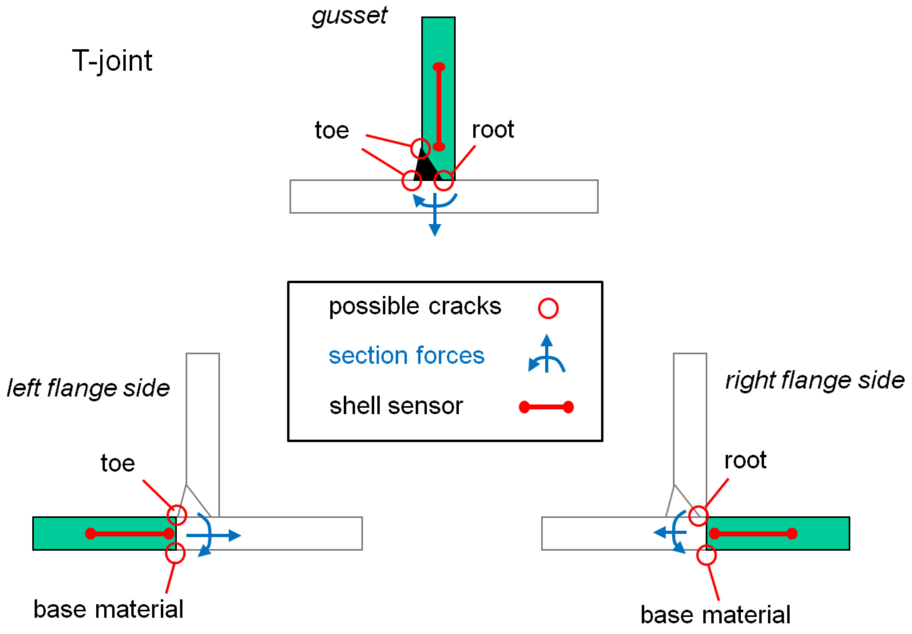


Figure 8: Embedded shell sensor elements, T-joint

4: Comparison of results

4.1 Test case 1: flange with gusset

The flange (thickness 15mm) is loaded by 100 MPa nominal stress, symmetry is used on the right side. In Figure 9 results for two different mesh sizes are shown. In the linear elastic formulation we face a stress singularity at the corner. The peak stress is strongly mesh dependent, as can be seen on the max. legend value in Figure 9.

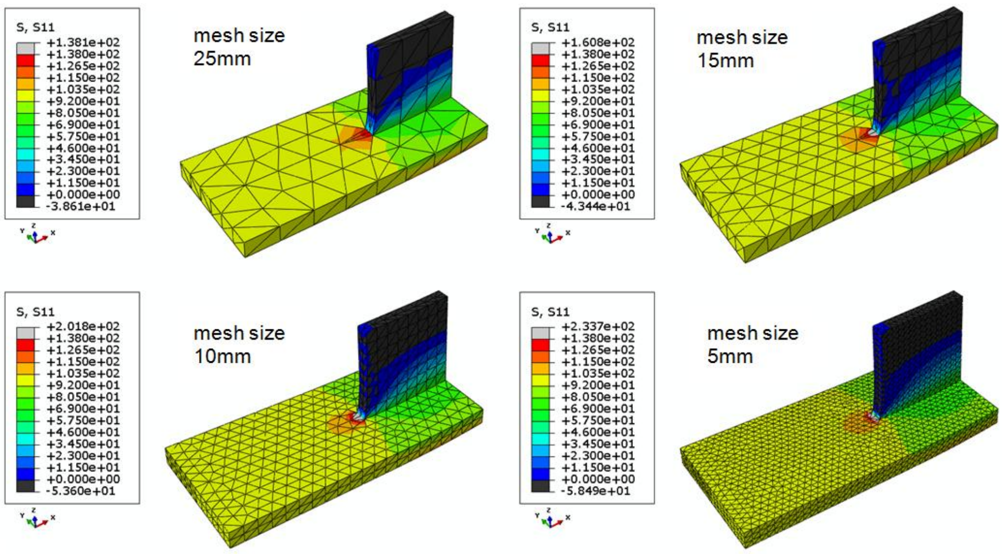


Figure 9: Flange with gusset, results for different mesh sizes

Figure 10 shows a comparison of structural hot spot stresses and nominal stresses recovered from different procedures:

Results a.) through d.) are calculated using displacement fields of the C3D10 elements in Figure 9 and the embedded sensor.

Solutions e.), f.) and g.) were found with the Abaqus shell elements S4R and S8R at different mesh sizes using the IIW extrapolation scheme on the shell element stress results. The result of the S4R element is close to the nominal stress.

Case h.) was found using Abaqus/FE-Safe in combination with a fine structured hexahedron mesh, based on the procedure of (Dong, 2006).

The values of cases i.) and j.) were found by linear extrapolation of surface stresses, which were calculated in the IIW reference points by interpolation within the C3D10 elements.

The real nominal stress is 100MPa (boundary condition). The correct structural hot spot stress is taken to be 122 MPa based on case h) with its structured and fine mesh.

Both stress measures are captured quite accurately by the new approach (case a-d) using simple tetrahedron meshes. This is rather independent of the mesh size up to mesh sizes of about 2.5 times wall thickness!

A simple stress extrapolation based on the same FEM tetrahedron results fails to predict the correct structural hot spot stress (case i-j).

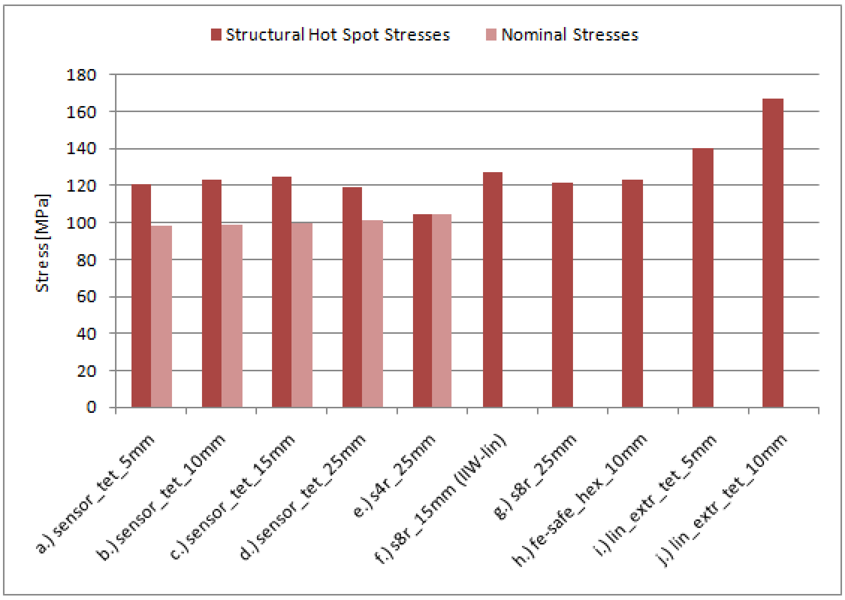


Figure 10: Comparison of different stress results for a flange with gusset

4.2 Test case 2: doubling plate

Figure 11 shows an embedded shell sensor for the extraction of structural stresses for a doubling plate (base plate thickness 10mm, doubling plate 8mm). The deformed structure and stresses were shown in Figure 6.

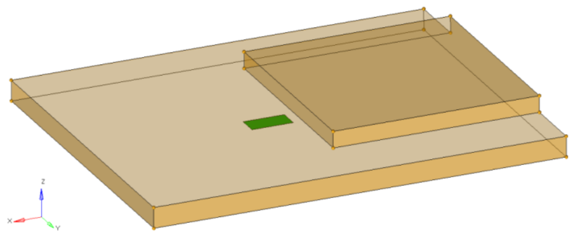


Figure 11: Shell sensor element near toe

Figure 12 shows once more a comparison of structural hot spot stresses and nominal stresses recovered from different procedures. These procedures are exactly the same as in test case 1 and described there.

The real nominal stress is 100MPa (boundary condition). The reference value for the structural hot spot stress is taken to be 170 MPa based on case h).

The new approach (case a-d) underestimates the reference stress slightly by 4% to 10% depending on the mesh size. Using large elements leads to an increased stiffness, smaller deflections and strains. This results directly in lower structural hot spot stresses.

Also for this test case the new approach (case a-d) is almost independent of the mesh size up to an element length of about 2.5 times wall thickness, while being accurate enough for the assessment of the structure!

A simple stress extrapolation based on the same FEM tetrahedron results in considerably higher stresses than the reference value (case i-j).

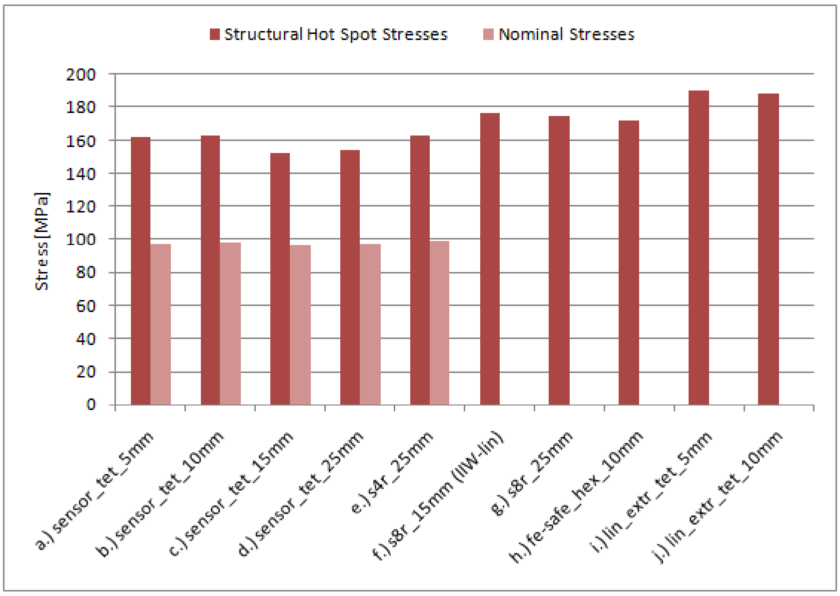


Figure 12: Comparison of different stress results for a doubling plate

4.3 Test case 3: welded structure

Figure 13 shows von Mises stresses for a complex welded structure modeled with C3D20 and C3D10 elements. The assembly has been created meshing the imported parts individually and using only TIE constraints to “weld” the parts together. The meshes at the interfaces are therefore non conformal. The mesh size is around wall thickness (testing of coarser meshes is in progress).

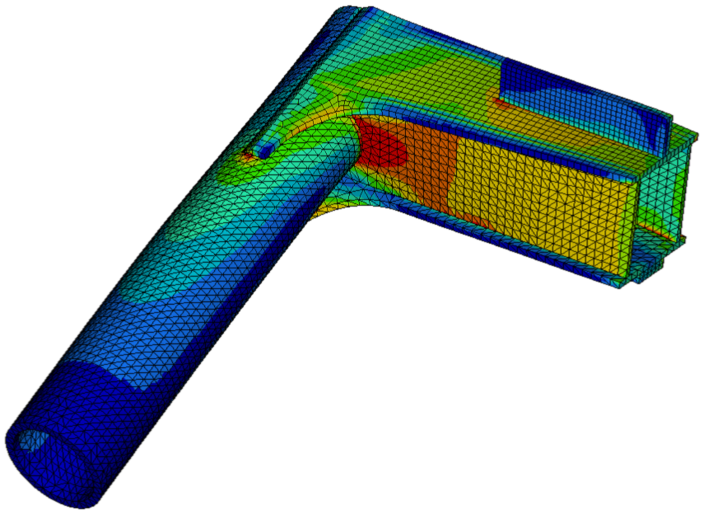


Figure 13: Solid model of welded assembly

Figure 14 shows the assembly in its shell representation. The element type is S4R and the mesh size is close to two times the wall thickness.

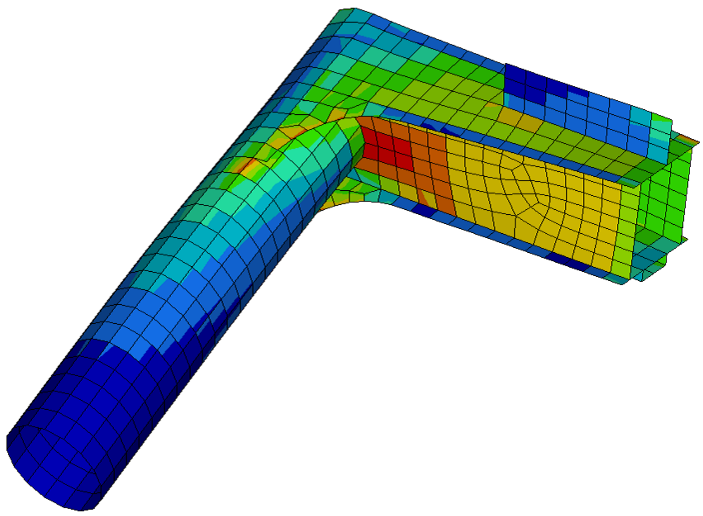


Figure 14: Shell model of welded assembly

Usually a large number of sensor elements are placed along the welds. Figure 15 shows a typical arrangement of sensor elements for three different joint.

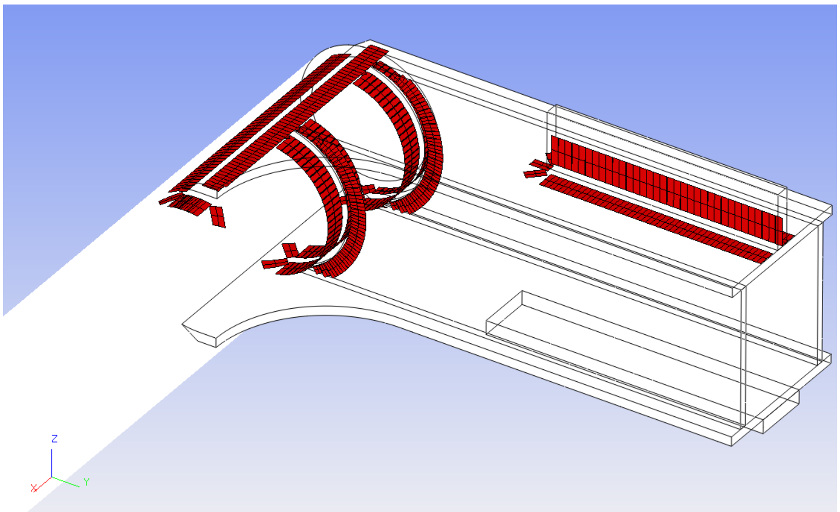


Figure 15: Typical arrangement of sensor elements

Figure 16 depicts the placement of selected most critical sensors used to evaluate the structural hot spot stresses from the results of the solid mesh. The sensor length is two times the local plate thickness.

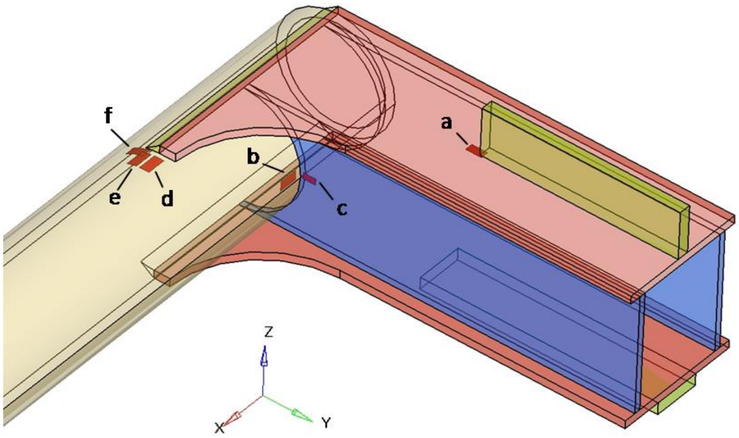
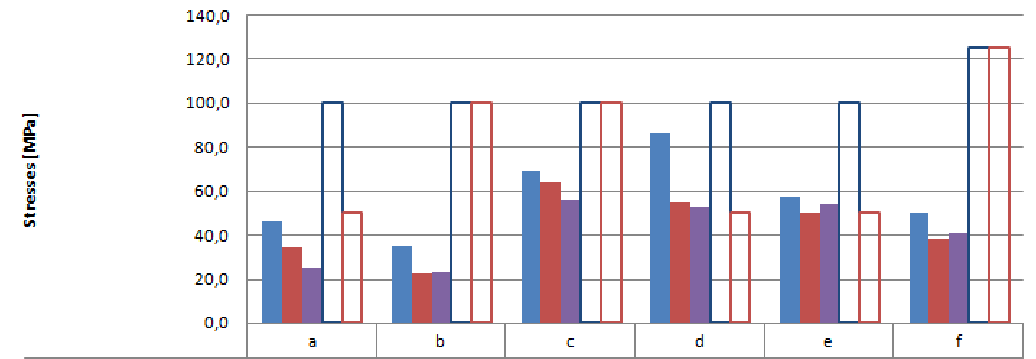


Figure 16: Selected sensor elements

Figure 17 lists stress results from both models together with the permissible FAT stress values taken from the IIW (Hobbacher, 2007). ‘Mean solid stresses’ are through-thickness linearised stresses taken from the sensor center without in-plane extrapolation. These stress values show good agreement with the shell results.



	a	b	c	d	e	f
structural hot spot stress	46,2	34,8	69,0	86,0	57,2	50,2
mean solid stress	34,0	22,2	63,7	54,4	50,2	38,4
shell center stress	25,0	22,8	55,9	52,9	54,1	40,6
FAT, structural hot spot	100,0	100,0	100,0	100,0	100,0	125,0
FAT, nominal	50,0	100,0	100,0	50,0	50,0	125,0

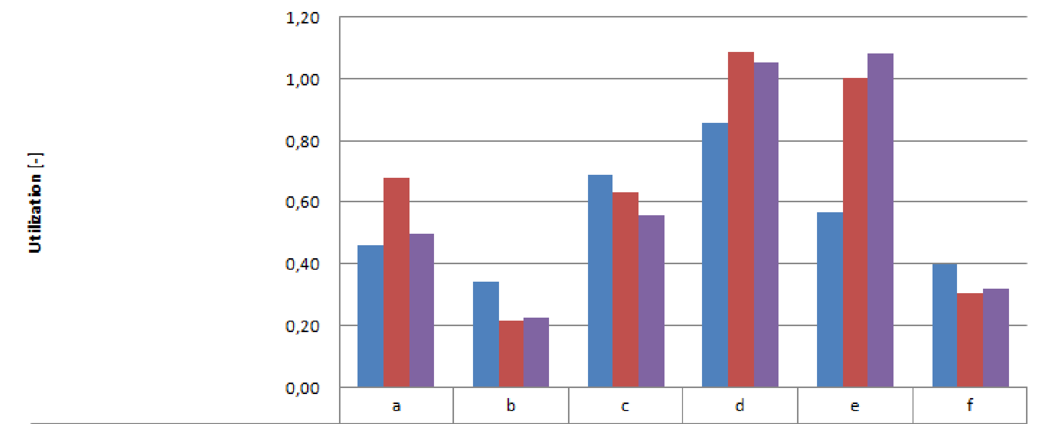
Figure 17: Stresses at different positions of welded assembly

For some positions (a, d, e) the permissible FAT hot spot stresses are twice as high as for the nominal stress approach.

Dividing the stresses through the permissible FAT values given in Figure 17 leads to utilization factors for each position (a-f), assuming no additional safety factors and a simple constant amplitude loading with two million cycles. The permissible stresses are the same for mean solid and shell approach because both represent stresses at a distance of approximately the wall thickness away from the weld toe. Therefore also the utilization factors, red and magenta bars in Figure 18, show good agreement since already the stresses showed little differences.

The only exception can be seen for position a, which has a stress singularity and shows mesh dependant results. For this position the structural hot spot approach is more reliable giving lower utilization ratios than the other methods (see also 4.1).

For the positions b and c the utilization factors for the structural hot spot method are higher than for the others, since the high local shear stresses acting at the weld toe are taken into account.



	a	b	c	d	e	f
utilization, structural hot spot stress	0,46	0,35	0,69	0,86	0,57	0,40
utilization, mean solid stress	0,68	0,22	0,64	1,09	1,00	0,31
utilization, shell center stress	0,50	0,23	0,56	1,06	1,08	0,32

Figure 18: Utilization factors at different positions of welded assembly

The results of Figure 18 show significant differences for the structural hot spot approach compared to the mean solid or the shell center method. The commonly used fatigue assessment approach based on shell results strongly depends on the selection of the FAT class and mesh size and can be non conservative depending on the weld details. In one region the mesh might be to fine partly, resolving structural stresses within the nominal stress approach (positions a, d, e). In other regions the mesh is not fine enough to capture the relevant stress level (b, c).

Using structural hot spot stresses simplifies the choice of FAT classes since the structural behavior is fully resolved by the Finite Element model. Additionally the sensor elements loosen the strong restrictions on element type and size usually involved with structural stress approaches.

Overall this new approach has the potential for significant time and cost savings within fatigue assessment processes, since no mid surface extraction and modeling is needed and the displacement field of the solid model can be directly used to derive structural hot spot stresses.

5: CONCLUSION

A novel method for calculating structural hot spot stresses from solid meshes is presented. This new embedded sensor element method shows significant advantages for the use in complex welded structures as compared to other solid model fatigue assessment methods, especially the use of coarser tetrahedron meshes. The only constraint on the solid mesh is a sufficient element size for the resolution of the displacement field around the welds.

Test cases show a good agreement with results obtained by procedures from literature.

The new method enables significant time and cost saving in the simulation cycle of fatigue assessment of welded structures by avoiding the common approach of mid-surface shell modeling of welded structures (time costly, etc.).

6: ACKNOWLEDGMENT

The work has been partially funded by ZIT Zentrum für Innovation und Technologie, Austria.

REFERENCES

Dong, P., 2006. *A Structural Stress Definition and Numerical Implementation for Fatigue Analysis of Welded Joints*, International Journal of Fatigue, 25, pp 359–369

Fricke,W. and Kahl, A., 2005. *Comparison of Different Structural Stress Approaches for Fatigue Assessment of Welded Ship Structures*, Marine Structures, 18, pp 473–488

Hobbacher, A., 2007. *Recommendations for Fatigue Design of Welded Joints and Components*, International Institute of Welding

Löffler, P., 2012. *Modellstudien für die Entwicklung eines universellen Strukturspannungskonzepte* (in german), Diploma Thesis, FH Oberösterreich