HOW TO ASSESS BIW STRUCTURES FROM THE FATIGUE POINT OF VIEW

Markus Kaltenböck, Dannbauer H., Hübsch W., Puchner K., Fischmeister S.
MAGNA POWERTRAIN
Engineering Center Steyr GmbH & Co KG (ECS), Structural Analysis Department
Steyrer Strasse 32, 4300 St. Valentin, Austria
Tel.: +43 (0) 7435 501 2324
Email: M.Kaltenboeck@ecs.steyr.com

KEYWORDS – Fatigue, Finite Elements, FEMFAT, Seam Welds, Spot Welds

ABSTRACT – For lightweight automotive body structures, the stiffness and the fatigue behaviour is massive influenced by the properties of its joints. The used joining technology, the number and the distribution of the joints are of high importance to derive well engineered and cost effective structures.
Numerical simulations, mainly based on the Finite Element Method (FEM) were implemented in the development process to reduce mass, increase body stiffness, optimize the crash and NVH behavior as well as to derive well balanced fatigue characteristics. Beside possible shorter time to market and lower costs these quite new technologies lead to optimized usage of material and production resources.

In the area of fatigue simulation there are some necessary specifics of the FE-model to enable such kind of investigations. On the one hand the whole durability analysis process has to be quick to enable efficient variants assessments. On the other hand the various possible joining technologies (like spot welds, seam welds, adhesives, self piercing rivets, flow drills,…) with its individual behavior regarding fatigue, leads to specialized modeling necessities for each type of connection. Therefore ANSA provides several possibilities to set up a FE-mesh ready to do a subsequent fatigue assessment with the ECS software FEMFAT.

This lecture shall represent the typical process in the area of BIW fatigue assessment. Starting from load time history preparation by a combination of simplified measurement and so called "Virtual Iteration", the specific set up and details of the FE-model and the subsequent fatigue analysis will be shown. The focus will be put on the joining technology representation in the FE-model as well as the possibilities in ANSA to optimize the respective pre-processing task.

Therefore the lecture shows how ANSA supports the engineer to derive the specialized models for spot welds as well as for seam welds. To cover the whole simulation process some more information about fatigue assessment of the different joining types will complete the picture of the process.

TECHNICAL PAPER

1   GENERAL EXAMINATION OF A BODY STRUCTURE DEVELOPMENT

The increasing globalization of the markets and the growing competition in the last decades resulted in a drastic shortening of the product life cycles. Since 1991, the average product development in mechanical components engineering e.g. was reduced from thirty to twelve months [1, 2]. This situation forces the enterprise to shorten the development time and to decrease product development costs. It leads to a conflict situation in which the three aspects development time, development costs and quality assurance interact closely.
The use of modern software such as simulation tools for example, enables finding possible counter measures. With their help, comprehensive product knowledge can be acquired at an earlier stage of the development process and therefore cost-intensive errors in later phases can be prevented and prototype cycles can be reduced.

Developments of body in white (BIW) structures are mainly influenced by 3 performance disciplines (crash, NVH, durability). For all these disciplines CAE methods already exist, mainly based on the Finite Element Method (FEM). All these simulation tasks are state of the art to reduce and avoid very expensive prototypes and tests. But only with reliable simulation results these savings will become reality.
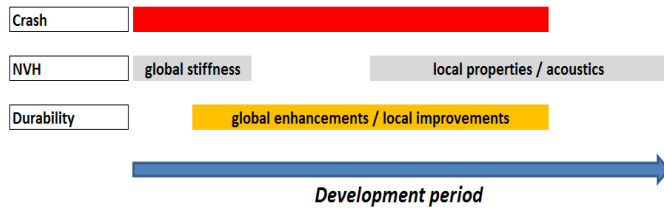
Multidisciplinary, automated optimization is a goal, but is currently practically not achieved due to the highly complex tasks and diverse requirements involved. Nevertheless, simultaneous engineering is necessary in any discipline where creative design optimization presents the greatest potential for improvements.

Figure 1: CAE time line of various disciplines in BIW engineering

This papers is focused on the durability assessments of body in white (=BIW) structures as well as on chassis parts.

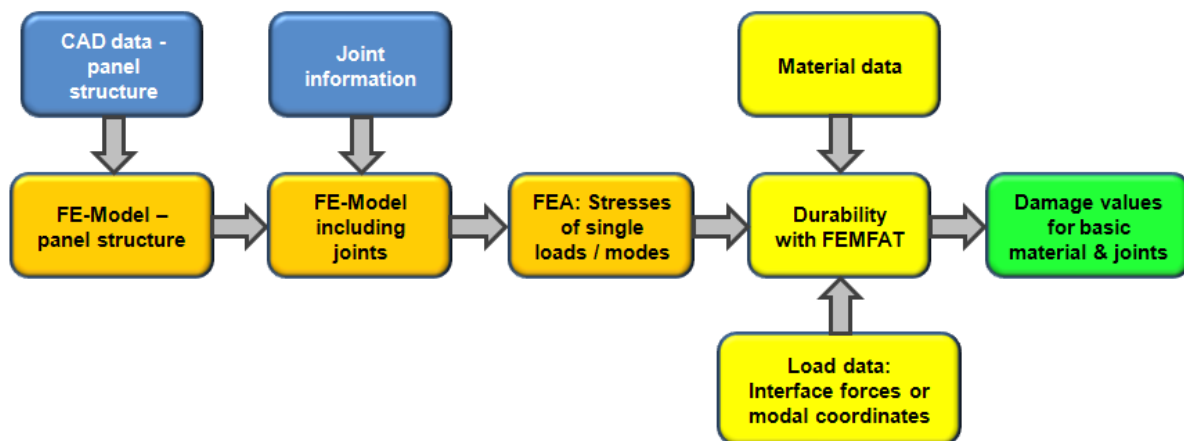Following flow chart represents the major process steps for fatigue investigations.

Figure 2: Durability Assessment Flow Chart

Typically after the first step – individual meshing of the panels itself - the joint information must be included. A detailed look onto the various possible joining technologies (like spot welds, seam welds, structural adhesives, self piercing rivets, flow drills,…) with its individual behavior regarding fatigue, leads to specialized modeling necessities for each type of connection. Since ANSA is specialized in set up of FE-models for complex BIW structures there are several possibilities available introducing these connections automatically or semi-automatically.

Simultaneous to the FE model preparation the load history input must be available to be used in the subsequent fatigue analysis. To derive these data normally measurement on reference vehicles will be done. Most OEM's use the expensive wheel force transducers (WFT). As an alternative method the so called "Virtual Iteration" (developed by ECS) can be used to examine these load data in combination with simplified measurements. Especially for truck cabs where the direct measurement of the cab suspension loads is very expensive, this process is a good opportunity to derive accurate load data via simulation. Also the transfer of loads from a measured previous vehicle to the new one is a helpful opportunity of this simu-

lation technology. Detailed explanations on this topic can be found in the paper "Integrating virtual test methods and physical testing to assure accuracy" [8].

2 SPOT WELDS AS THE MAJOR JOINING TYPE FOR BIW STRUCTURE

Besides the meshing of sheet metal panels and the correct computation method one main topic in the simulation field of BIW is the modelling of the joints. With a close look onto the details there are a lot of different possibilities to represent and simulate different joining technologies. Especially for spot welds several software houses have developed various possibilities to include the behaviour of the spot welds, more or less detailed.

2.1 Spot Welds for Fatigue Simulation

The main goal of the spot weld model in the field of fatigue simulation using the Finite Element Method is to connect the shell elements of the metal sheets and to lead to reasonable results regarding local stresses.

In the first time one was seduced to use one-dimensional node to node connecting elements (beams or rigid elements) to realize welding spots. The problems with mesh density depending stress distribution or high sensibility of the beam forces regarding little inclination deviations of the connector elements led to the so called "second generation of spot weld models". These modeling techniques with the advantage of mesh independent sheet connections combine more or less the possibilities of quicker mesh generation with more accurate results.

Some examples of the most common connections used nowadays:

- Nastran CWELD: Created by MSC together with Volkswagen
- Solid & Coupling Elements: Created by CDH AG. Combination of solid and coupling elements
- Abaqus Fastener: Developed by Abaqus
- FEMFAT nuggets: Developed by the Engineering Center Steyr / MAGNA Powertrain

For the fatigue assessment of spot welds with the software package FEMFAT (developed and worldwide distributed by the Engineering Center Steyr a company of the MAGNA POWERTRAIN group) either the section force of the connector element (**force based concept**) or the stresses in a well-defined ring of elements around the joining element (**stress based concept**) can be used.

2.1.1 Fatigue Simulation Possibilities of Spot Welds in FEMFAT

The operating strength of sheet metal parts of bodies is strongly influenced by the properties of their joints. The used joining technology, the design and the position of the welding seams, as well as the number and the location of the spot joints have a significant technical and economical impact. This becomes clear from the fact that in fatigue testing of car bodies more than 90% of the cracks originate from joints [3, 4, 5]. This is mainly due to the considerably lower dynamic loading capability of welding seams and spot joints compared to the base material.

Possibilities to analyze the durability of spot welded structures with FEMFAT

- *Stress-based assessment of spot joints with FEMFAT nuggets*

More or less independently of the fineness of the shell element basic FE-model new meshes are automatically applied to the local areas around the spot joints before carrying out the FEM analysis. The resulting nugget-like arrangement of elements forms the basis of uniform evaluation quality. In the course of this local enhancement of the FEM mesh, the local rigidities are adjusted to the used joining technique (spots welds or self piercing rivets).
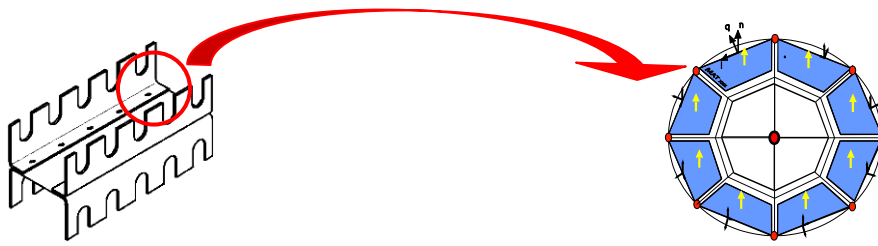
Figure 3: H-Specimen for tension peel loading with 10 spot joints and a detailed model of a FEMFAT spot nugget

The stresses in the shell elements around the spot joint axis are evaluated. Depending on the type of local loading, tension/compression, shear or, more usually, a mixture loading, different S/N curves are used for the damage accumulation. The shape and sensitivities of these S/N curves were derived from tests on selected test bodies (H - samples and DC - samples) and are provided in a database.
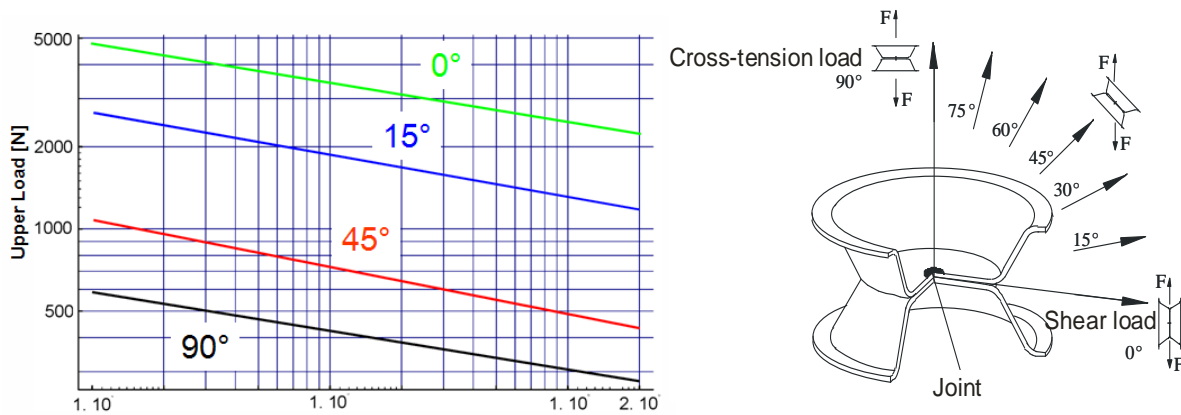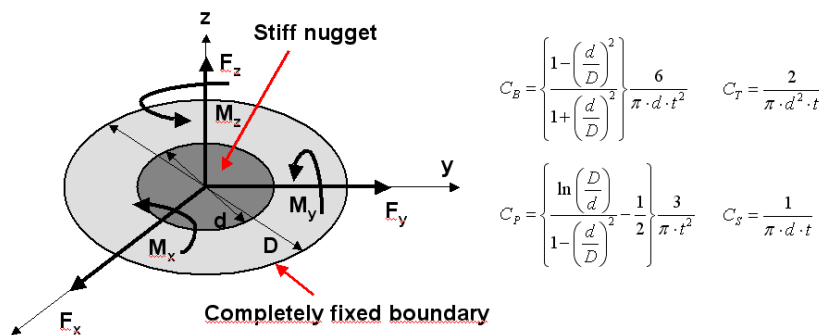

Figure 4: Local spot weld S-N curves implemented in the database of FEMFAT; evaluated with a comparison of simulation and test results of DC-specimen and H-specimen

- *Force-based assessment of spot joints in FEMFAT*

In order to be able to assess spot welds modeled as CWELD's or Solid & Coupling elements an additional assessment method was implemented in FEMFAT.

$$C_B = \left\{ \frac{1 - \left(\frac{d}{D}\right)^2}{1 + \left(\frac{d}{D}\right)^2} \right\} \frac{6}{\pi \cdot d \cdot t^2} \qquad C_T = \frac{2}{\pi \cdot d^2 \cdot t}$$

$$C_P = \left\{ \frac{\ln\left(\frac{D}{d}\right)}{1 - \left(\frac{d}{D}\right)^2} - \frac{1}{2} \right\} \frac{3}{\pi \cdot t^2} \qquad C_S = \frac{1}{\pi \cdot d \cdot t}$$

$$\sigma_{rsum} = C_P \cdot F_z + C_B \cdot \left(-M_y \cdot \cos\theta + M_x \cdot \sin\theta\right) - C_S \cdot \left(F_x \cdot \cos\theta + F_y \cdot \sin\theta\right)$$

$$\sigma_{\theta sum} = \nu \cdot \sigma_{rsum}$$

$$\tau_{r\theta sum} = C_S \cdot \left(F_x \cdot \sin\theta - F_y \cdot \cos\theta\right) - C_T \cdot M_T$$

Figure 5: Calculated stresses due to internal forces and moments of the connection element
The internal forces and moments occurring in connecting elements can be used as the basis for an evaluation of the operational strength of spot welds without any alteration of the mesh. The used algorithm accords to the determination of nugget stresses by Nakahara,

4

Takahashi, Kawamoto, Fujimoto and Tomioka suggested in the SAE paper 2000-01-0779 "Method of Fatigue Life Estimation for Spot-Welded Structures" [6].

2.1.2 Introduction in FEMFAT Nuggets

Although, as mentioned, common connections like "Nastran CWELD" or "Solid & Coupling" can be evaluated regarding fatigue with FEMFAT using the "force based concept", the most accurate possibility is the usage of the so called "FEMFAT nuggets".
It is a matter of fact that the element quality close to the spot weld influences the stiffness, the stresses and of course the internal forces of the spot weld connection element. Without local remeshing, as it is a part of a durability analysis using FEMFAT nuggets, the element quality will influence the durability results significantly. I.e. it is a very quick possibility using rough meshes, as they are useful for NVH investigations, in combination with the force based concept to assess the durability behaviour of BIW structures. But it is a matter of fact that the fatigue results can vary due to the influence of the base mesh quality to the connection element interface forces.

Possibilities to derive FEMFAT nuggets in the FE-model:

- *FEMFAT Spot Remesher*

To generate these connections a pre-processor is available within FEMFAT SPOT which serves to automatically modify the local FE-mesh around the spot welding. The result yields a steady circular element pattern around the spot joint in the magnitude of the diameter of the weld diameter.
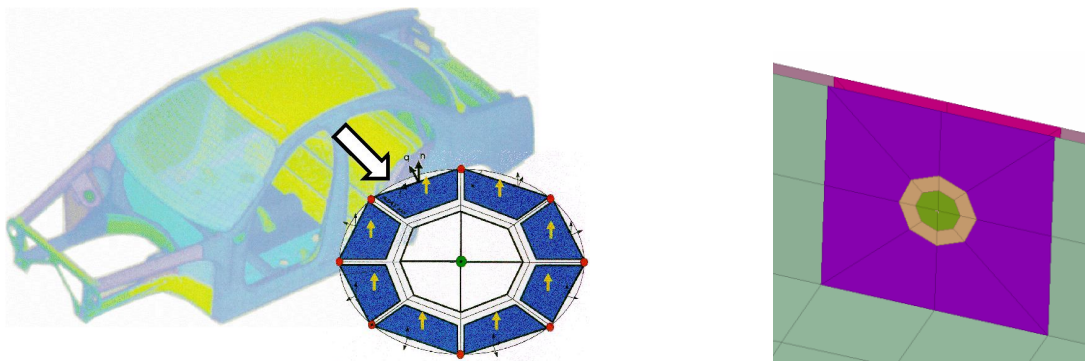

Figure 6: Automatically pre-processed local spot joint area of a body in white structure

The main problem for the remeshing process using FEMFAT SPOT Remesher is that only the FE-model is the base for the nugget model implementation and for the connection of the nugget to the rest of the panel's mesh. I.e. there are no geometrical background information available (like CAD data). Especially in case of nugget implementation using rough base meshes some replacements will be faulty.

- *ANSA FEMFAT Spot Remesher*

To avoid necessary manual mesh adaption the strong meshing potentials of ANSA can nowadays substitute the usage of the FEMFAT Spot Remesher. Therefore ANSA has implemented the possibility to replace connection points by FEMFAT nuggets.
The availability of the ANSA feature to introduce a second element ring (Spider 2) has an additional advantage. Although the number of elements will increase, the dependency of the local stresses of the spot weld assessment elements to the base mesh roughness will decrease. Thus model driven differences of the fatigue results by replacing panels with a new meshed one will be reduced.
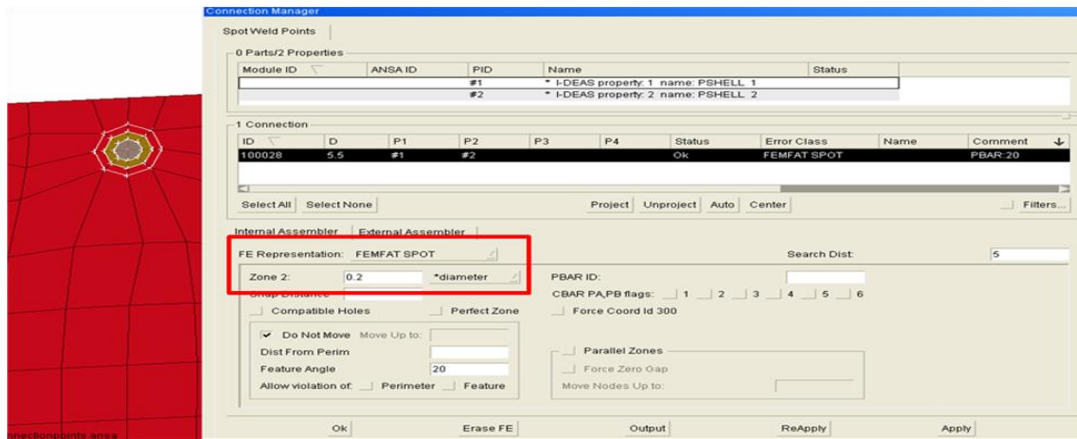
Figure 7: Screen shot of the connection manager of ANSA together with the results of the FEMFAT nugget

As mentioned above the ANSA Spot Remesher works quite independently of the quality of the base mesh. Even for rough meshed panels, often used in the area of NVH investigations to reduce analysis time, the resulting nuggets are represented in good quality as it can be seen in the next picture.
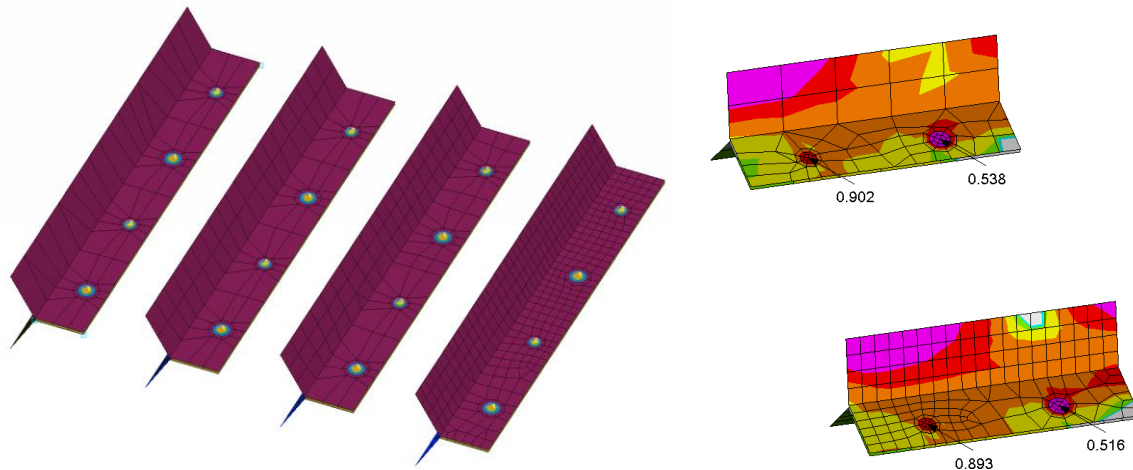

Figure 8: Remeshing results of ANSA replacing connection points by 2 rowed nuggets for different base meshes and according FEMFAT endurance safety results of the spot welds and the base material

First applications promise faster and more reliable fatigue results with lower scattering depending on the base mesh quality. Exemplarily remeshings on parts of real BIW structures confirm these statement as it can be seen in Figure 9.
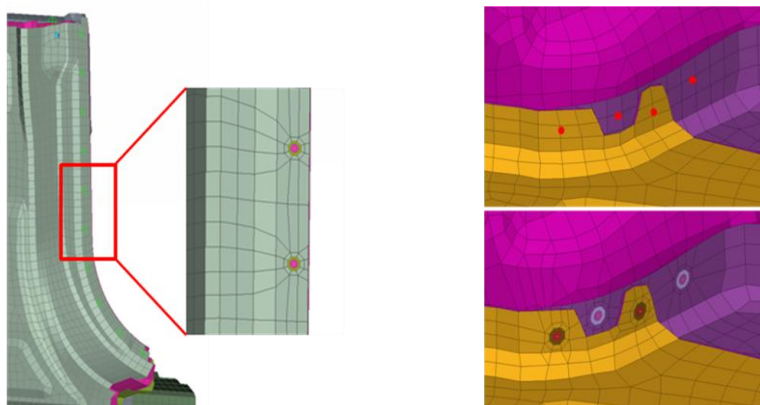

Figure 9: ANSA spot weld remeshing tested on real BIW structure parts

Of course since it is a rather new feature of ANSA some teething problems are unavoidable. In case the spot weld location is close to a panel's border the remeshing results can lead to unexpected problems. The next picture represents such kind of critical area. The Spider 2 feature leads to a second element row which is very close to the panel's edge. So ANSA has to decide either to use a quadrilateral element (highlighted in the picture) without changing the border line of the structure but with poor element quality or the usage of a triangle element by accepting a small deviation of the original border. An improvement in the meshing algorithm is desired where the allowed changing of the structure can be defined by the customer. Replacing the spot weld connectors of a base mesh without reference to the geometry, which is very common in our daily business, the shape of the original mesh is changed. For instance in the area of fillets as it can be seen in the next figure. An efficient user defined definition of the allowable shape deviation will be very helpful to derive a ready to use FE-model without any subsequent manual improvement work.
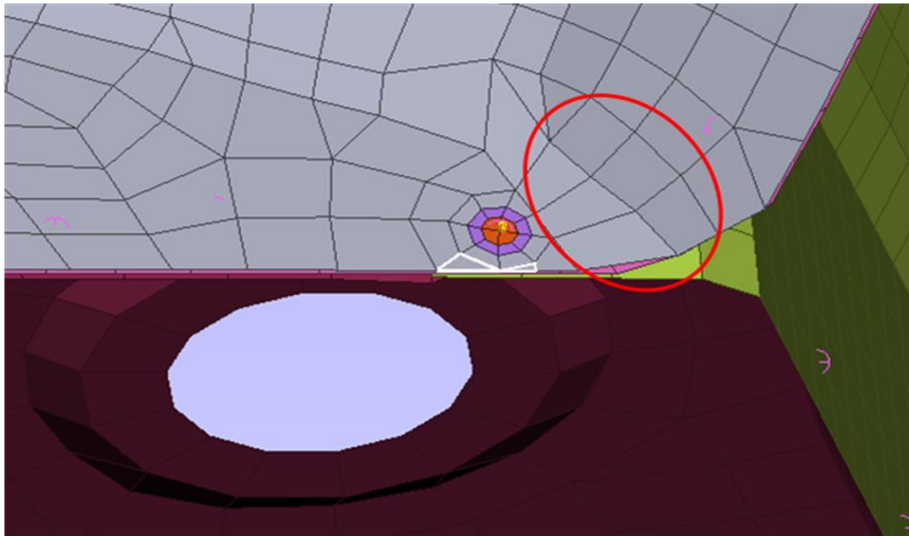

Figure 10: Example for problematic regions during spot weld remeshing

However the possibility of nearly automatic implementation of FEMFAT nuggets in addition of a second element row, which leads to more accurate stresses in the assessment elements, sounds to be the future process in the area of spot weld assessment.

3 SEAM WELDS ASSESSMENT AND IMPLEMENTATION INTO THE FE-MODEL

3.1 Introduction into the Fatigue Simulation of Seam Welds

Beside the spot weld connections seam welds are often used in BIW and chassis structures. The fatigue software FEMFAT gives the possibility to analyze the fatigue behaviour of complex welded components by the use of Finite Elements (FE) simulation results.
It is possible to assess both shell models and combined shell/solid models.
After finishing the FE-model set up the definition of the weld seam within the FE-model is very easy by the use of special node attributes (e.g. local coordinate system labels) and material labels according FEMFAT WELD guidelines.
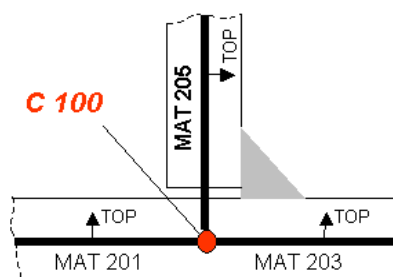

Figure 11: Example of FEMFAT WELD modeling guidelines

Both in the middle of the weld seam and at the weld seam beginning and end the root and toe can be assessed.

A simple FE shell model without consideration of weld seam details is sufficient, to make lifetime predictions. This makes it possible to evaluate welds and base material within one analysis loop in complicate welded structures like subframes, welded links or whole BIW. Weld seam specific information like notch factors and SN curves for root and toe are stored for each joint and seam type in a database. These parameters were acquired from submodels according Radaj, from tests or from different publications and standards.
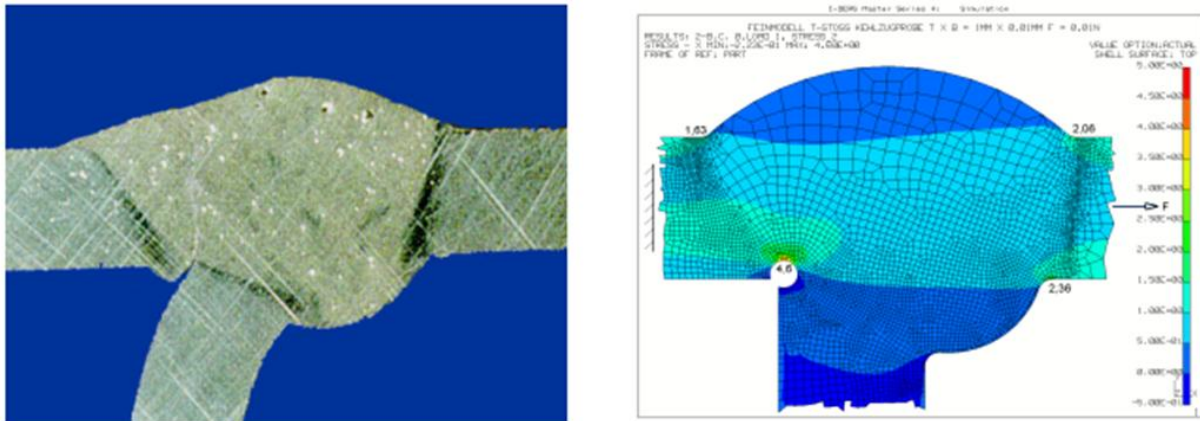

Figure 12: Submodel according Radaj for calculation of notch stresses

To avoid making submodels for every welding in the component, this was done once for the most common joint types and seam shapes. The notch factors for the most important load cases (tension, bending and load flow over web plate) were stored in a database. For the assessment of the welded components these notch factors will be used to approximate the notch stresses on the basis of the structural stresses from the FE-analysis.

The notch stress (based on 1mm notch radius) can be assessed using the master S/N curve according Koettgen, Olivier and Seeger (R1MS concept).

3.2   How to tell FEMFAT which Seam Weld Type shall be used

In previous days iime consuming work for engineers was to define the type of seam weld in a way understandable for FEMFAT. I.e. the set up was performed by manual definition of element material labels and local coordinate system labels of the seam weld nodes. Nowadays this job can be done very comfortable by new features of the **FEMFAT Visualizer** – the postprocessor module of FEMFAT.
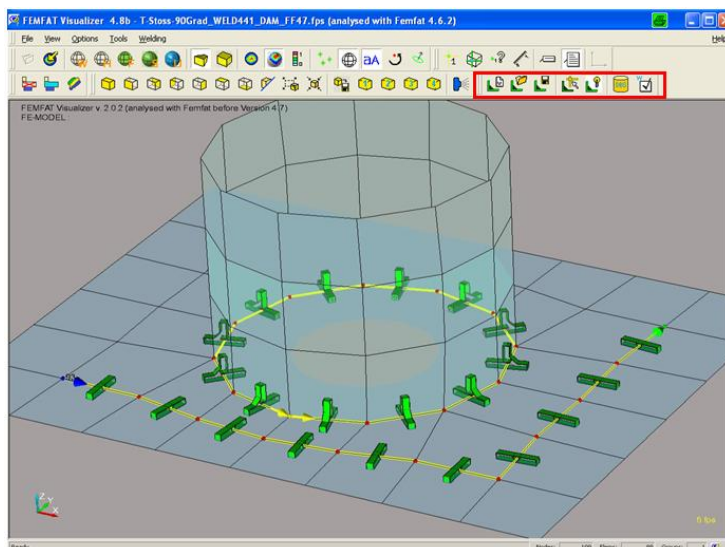

Figure 13: FEMFAT Visualizer seam weld illustration

This quite new possibility accelerates and simplifies the seam weld definition exorbitant.

The route of the weld can be specified with only a few mouse clicks on the imported FEM geometry. The joint type (T-joint, lap joint, etc.) is automatically recognized, only details need to be added – with visual support. The detailed weld geometry is displayed within the FEM model (see Figure 14) allowing rapid and unambiguous checking.
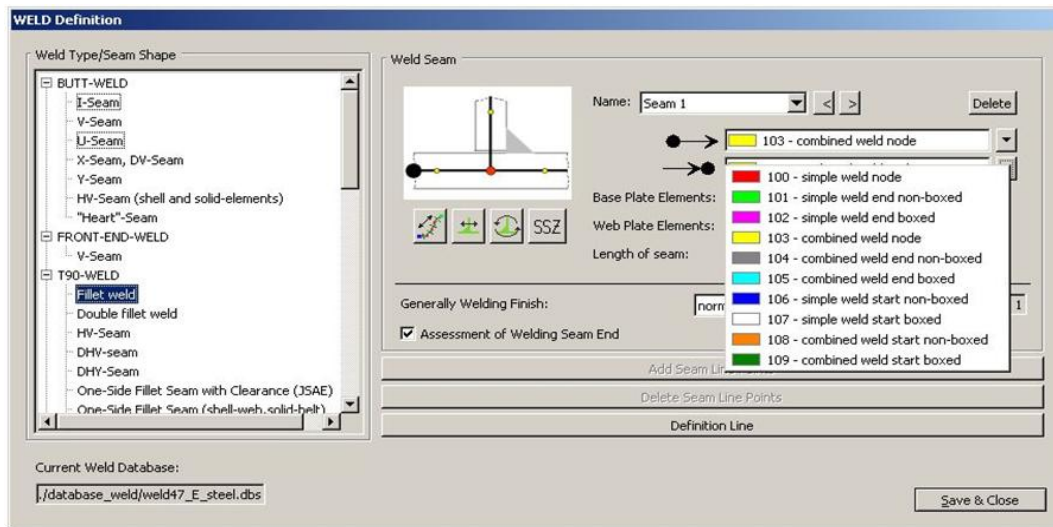

Figure 14: FEMFAT Visualizer GUI to define seam welds

Definition of seam weldlines:

> • Select the weld start node – possible weld lines are marked white
> • Select a node or a line segment on one of the lines
> • New weld lines are displayed to continue definition
> • Complete by using „End of weld definition" in the menu

The joint type, weld shape, weld side, weld direction, attributes for weld ends and beginnings (all with visual updating) can all be modified interactively after specifying the weld line. The defined weld information can be stored in a special file and can be imported for analysis or to continue the definition.

3.3   FE-Model Creation and Definition of Welded Structures with ANSA

3.3.1   Previous Way to create seam welded FE-models

As described in the previous item, with the FEMFAT Visualizer a quick definition of the seam welds of the complete FE-model is possible, but one problem can only be solved within the pre-processor – modeling the connection of the welded parts.
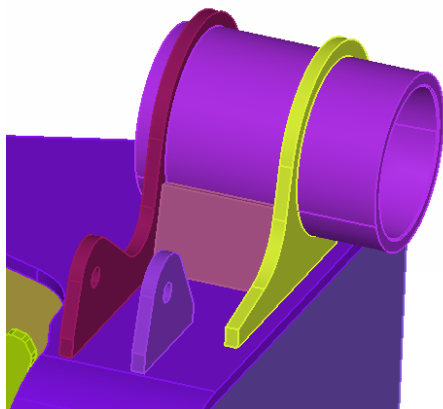
Normally the CAD data contain the panels as solids. In figure 15 an example of a typical 3D CAD data of welded structures can be seen.

Figure 15: Solid CAD data of welded structures

The FE-model shell structure is defined at the midsurfaces of the panels. I.e. after defining the midsurfaces there will be a gap between the parts.
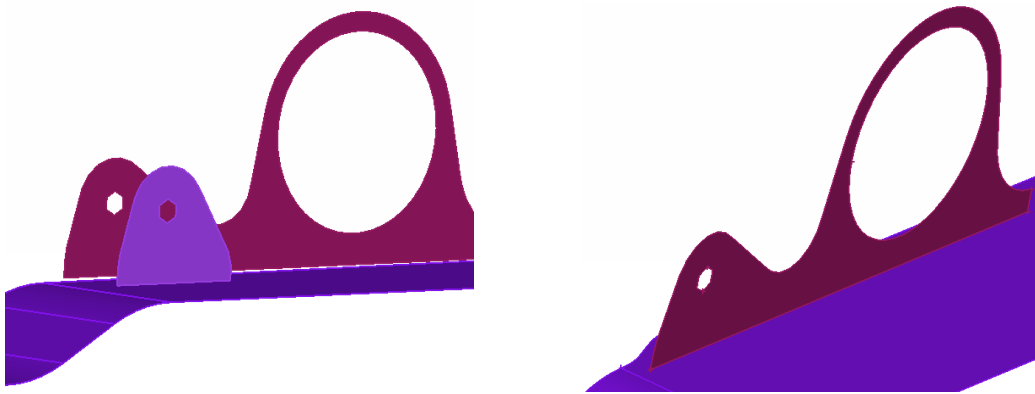

Figure 16: Midsurfaces without connection and intersection line

In previous times it was a long term manual task to enlarge the mesh surfaces to have an intersection curve between the surfaces of interest. Afterwards the joint surfaces had to be enlarged to have a connection between the parts ready to do the meshing.
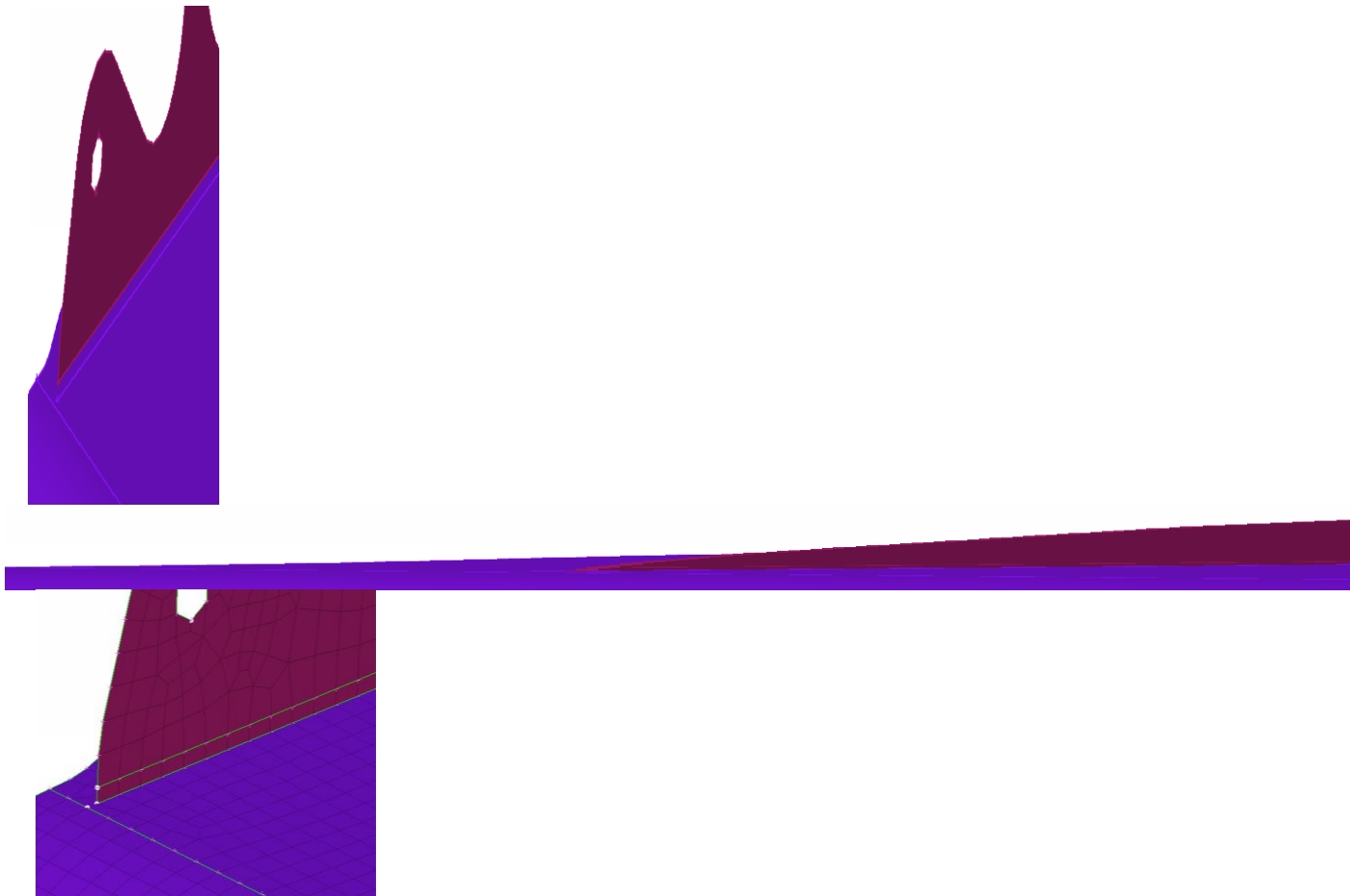

Figure 17: Intersection line creation, surface enlargement, FE mesh creation

3.3.2   Modern Method of Creation and Definition of seam welded FE-models

Nowadays ANSA provides good features to avoid manual work to create a FE-model of structures with seam welds based on CAD data available as solids.

By using the seam weld feature it is possible to semi-automatically extend the panels and define ready to analyze FE-models which fulfill the definition criteria of FEMFAT to assess the seam weld regarding fatigue.
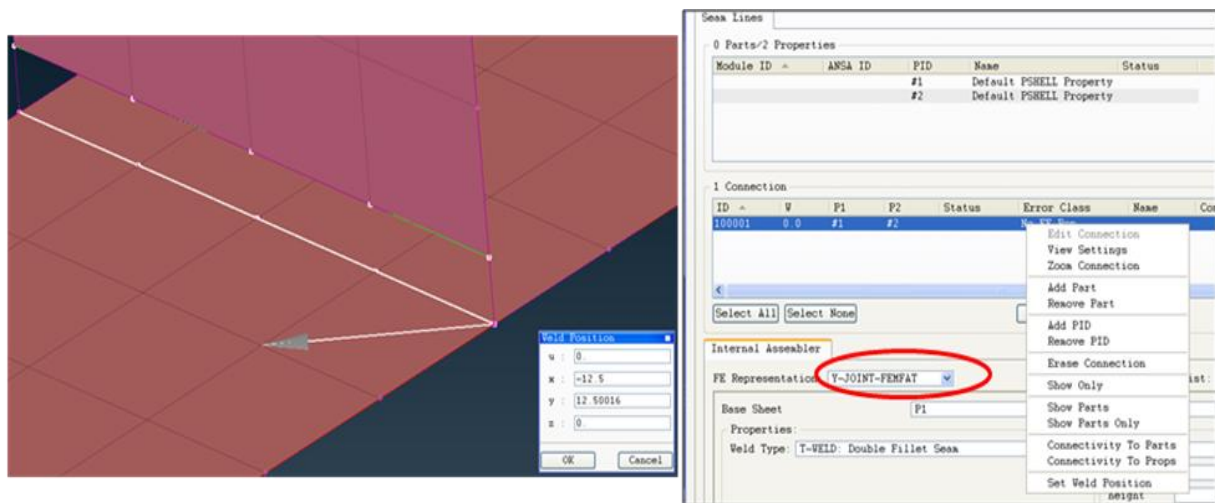
Figure 18: FEMFAT T-joint set up within ANSA

After finishing the set up the desired shell mesh including the FEMFAT seam weld definition will be created and can be used for FE analysis as well as for fatigue assessment.
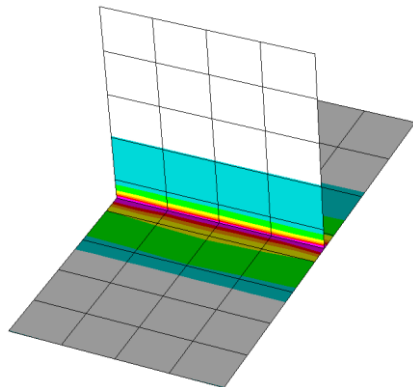
Figure 19: Fatigue results of the seam weld

Of course more complex seam welded structures can be prepared for the analysis and durability assessment by using this feature.

Although up to now only the most common joints are available this is one future feature which will reduce pre-processing time and strengthen the competitive power of ANSA.

Up to now generic examples were shown and it was explained how to derive the necessary FE-models and the definitions of the most common joining technologies in the area of BIW and chassis parts fatigue assessment. But of course these processes and technologies are proven with respect to the daily work performed in our house. The next chapter deals with some examples of fatigue assessments. It is a pity, but due to understandable confidentiality reasons only some selected investigations from the past can be represented.

4   EXAMPLES OF REAL STRUCTURES

The FEMFAT SPOT and WELD module was used for a lot of different assessments during real structure development. Beside the usage of FEMFAT SPOT at European and Japanese OEM's site many BIW and closure structures were investigated at ECS and compared as far as available to test results. Some example shall demonstrate the accuracy of the predicted critical fatigue results.

4.1 Spot Weld FEMFAT analysis results compared to test results

Example 1: Door analysis

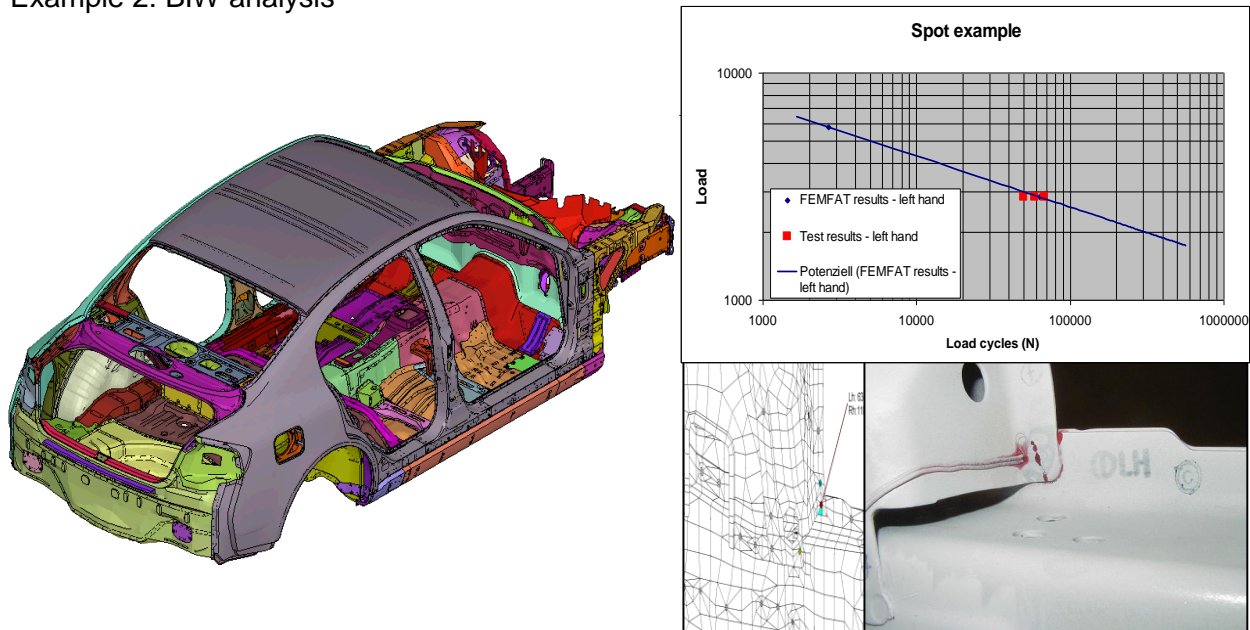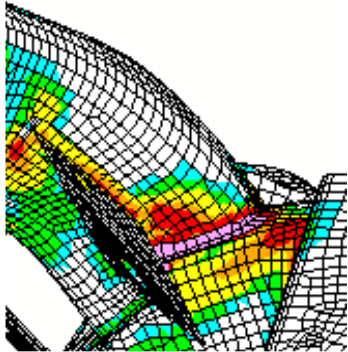| Fatigue Test	10.000 Cycles
FEMFAT	9.200 Cycles
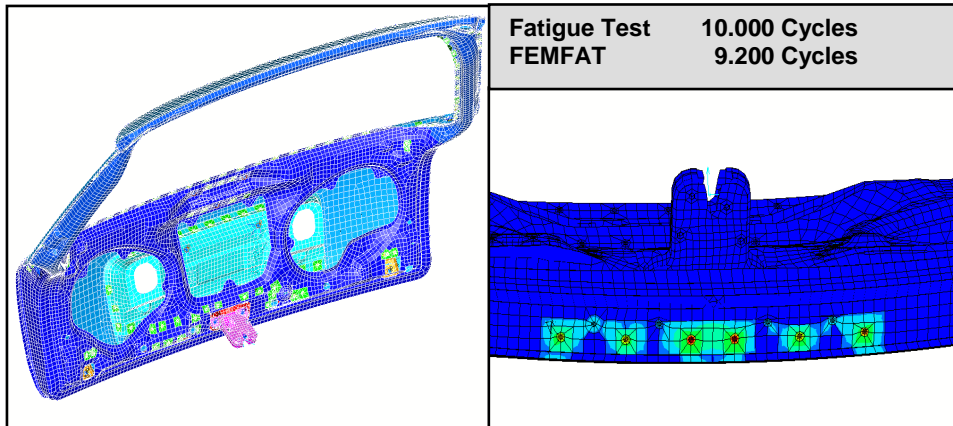
Example 2: BIW analysis

Figure 20: Examples for spot weld durability assessment of real structures

4.2 Seam Weld FEMFAT analysis results compared to test results

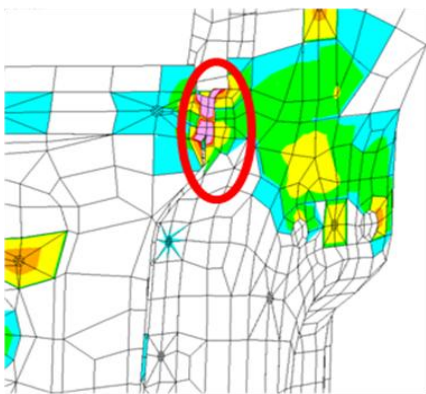Example 1: Subframe analysis

Example 2: BIW analysis

Figure 21: Examples for seam weld durability assessment of real structures

5 OUTLOOK

Besides the already mentioned symbiotic advantages of the tools ANSA and FEMFAT in the area of fatigue investigations of seam and spot welded structures like BIW and chassis parts some further development of our house shall be appointed under this paragraph. The new software product called **MAMBA**, with its first release in 2010, will enable to take into account the contact behavior between all relevant flanges. It is reasonable that this new approach, a co-simulation between FE-Solver, Multi-Body-System Simulation and FEMFAT will initiate a new era of fatigue simulation. Up to now the contact was either neglected or investigated in a simplified manner. With MAMBA all the stiffening effects or load flow changes due to contact situations can be taken into account. It is a matter of fact that the accuracy of the durability analyses will increase with this new approach.

A new module called **FEMFAT Frequency** will enable to assess stochastic excited structures doing FEMFAT multi-axial analysis using power spectra densities (PSD's) for several directions.

Robustness evaluation of seam welded structures will help to estimate the possible deviation of the fatigue results according to the variability of the production process. This new feature will be available in version FEMFAT 5 and will reduce production costs since the areas of

special accuracy during production (i.e. high sensible areas regarding production influences) can be detected and either redesigned in a very early development stage or especially controlled during production.

A new FEMFAT nugget model with more circumferential elements will be implemented soon to enable a more accurate stress evaluation. Hopefully it will be implemented in ANSA as well.

6   CONCLUSION

Methods for the modelling and for the durability assessment of spot joints and seam weld have been presented. The realization of the concepts within the user-friendly, reliable fatigue software FEMFAT allows a fast and efficient assessment of the operating strength of such structures. The usage and implementation in the development process leads to less tests, shorter development phases, higher quality and therefore less costs.
Two possibilities of fatigue prediction for spot welded structures are available. While the stress based concept requires a local modification of the mesh in the area of the weld spot, which can be obtained with an integrated spot remesher in FEMFAT or very powerful within ANSA, the force based evaluation uses the internal forces and moments of the connection elements and can be used for non congruent meshes. The results of the nugget model are less sensitive regarding basic mesh size and quality. Together with the possibility of the implementation of a second well defined element row using the ANSA Spider 2 option the accuracy of the local spot weld stresses and so, of course, the accuracy of the fatigue results can be increased.

Two possibilities to derive a ready to run FE-model of seam welded parts for FEA and the subsequent durability analysis were presented. On the one hand the FEMFAT Visulizer, a powerful post-processor for FEMFAT results, enables a quick definition of the seam welds with the advantage of visual inspection.
On the other hand ANSA improved the FE-model creation for midsurface based shell meshes with the opportunity to combine it with the definition of the seam weld type according to the FEMFAT specification.
Although there are ideas available to further improve, these features of the current ANSA version will simplify the daily work of the analysts and helps to reduce the possibility of errors.
The new developments in our house in combination with the quick reaction time of the ANSA development team leads to an efficient way to assess jointed panel structures more realistic than it was possible up to now.

But nevertheless a lot of further improvements are necessary to derive more and more precise fatigue results.

REFERENCES

[1] Fraunhofer IAO Studie 1996

[2] Fraunhofer IAO Studie 2000

[3] Gumpinger, J.; Hahn, O.; Korte, M; Kudrnac, P; Singh, S; Unger, B. (1997) "Computer Simulated Estimation of the Fatigue Behaviour and Stiffness of Spot Joints in Automotive Structures", Proceedings IBEC´97 – International Body Engineering Conference & Exposition, Stuttgart, Germany, September 1997.

[4] Singh, S; Schmid, G; Gao, S. (1991) " Leichtbau durch optimierte Fügetechnik", lecture and paper on the occasion of the DVM Meeting "Bauteil 91" at Berlin, Germany. Proceedings page 239-255

[5] Singh, S.; Schmid, G. (1993) "Dünnblechkonstruktionen und ihre Eigenschaften – Prüfverfahren und ihre Aussagekraft", Special Meeting of the SLV-Munich "Fügen von Aluminium im Dünnblechbereich", Ingolstadt, Germany, November 1993, lecture and paper, Proceedings page 17-26

[6] Y. Nakahara, M. Takahashi, A. Kawamoto, M. Fujimoto, N. Tomioka (2000) "Method of Fatigue Life Estimation for Spot-Welded Structures", SAE 2000-01-0779.

[7] Prof. Dr.-Ing. Ortwin Hahn, Dipl. Ing. Jan R. Kurzok, Dipl. Wirt. Ing. Andreas Rohde (1998) „Untersuchungen zur Übertragbarkeit von Kennwerten einer punktgeschweißten Einelementprobe auf Mehrelementprüfkörper und Bauteile"

[8] Dannbauer Helmut, Meise Matthias, Gattringer Otmar, Steinbatz, Michael (2006) „Integrating virtual test methods and physical testing to assure accuracy and reduce effort and time"; SAE 2006-01-3563.