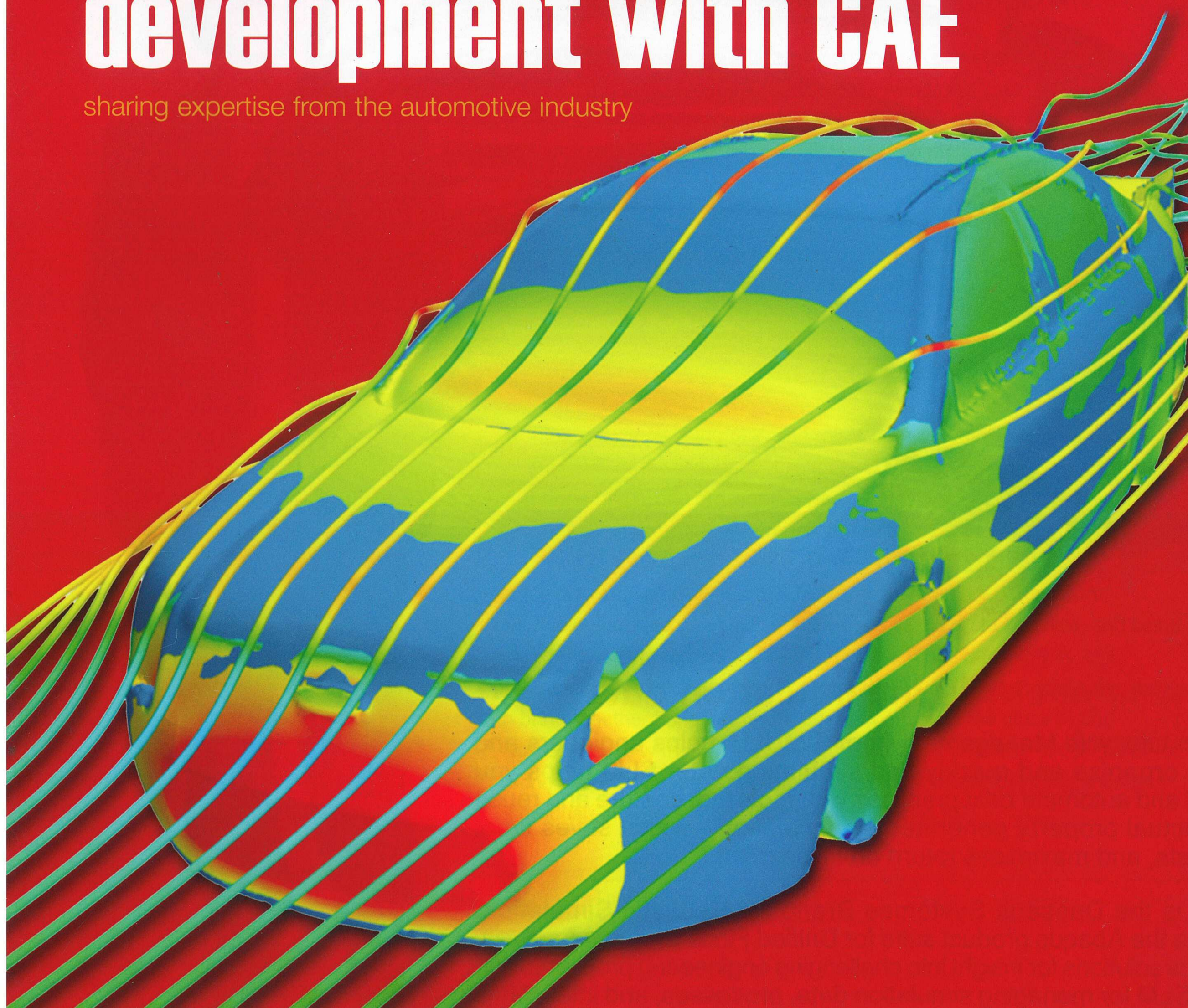
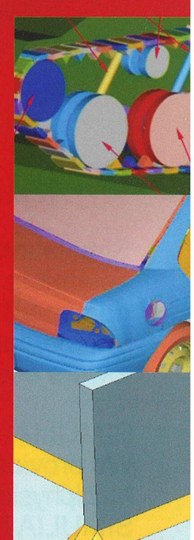
benchmark

the international magazine for engineering designers & analysts

improving product development with CAE

sharing expertise from the automotive industry





also in this issue . . .

Seam Weld Simulation | NAFEMS Regional Conferences

Simulating The Wiesel | Icons of CFD

april10

NAFEMS

Seam Weld Simulation

D. Turlier, P Klein, A. Wilhelm and M. De Azevedo. LOHR industrie, France

As a manufacturer of large welded-structure vehicles, the French company LOHR aims to optimize process cost. Seam welding, mainly fillet welds, is an important part of the manufacturing process. Fewer seams, throat size reduction, and intermittent welds are solutions for cost-effectiveness. But the strength of the assembly still needs to be ensured. In CAE, weld modelling is crucial for such detailed needs. LOHR Industry's "seam-sim" project consists first of elaborating and proving an innovative method where weld joints are represented by shell elements. Finally, a partial automation of the process from the Seam solid to the Simulation model is performed.

The most accurate way of analyzing welds consists of modelling the seam weld joint with solid elements. This method enables the evaluation of stresses within the weld joint. Unfortunately, this method requires a fine finite element model with a large amount of nodes. That is not suitable for studying a global vehicle structure. The purpose of this weld model investigation is to use a shell element model.

The first step of the model preparation consists of idealizing sheet metal bodies to mid surfaces (see Figure 1).

For static strength assessment, the stress through the weld throat section has to be evaluated. The weld shell element will have the throat minimum size assigned as thickness (Figure 2). An equivalent stress is calculated at point A using Eurocode 3 formulas [1]:

$$\sigma_{eq} = \sqrt{\sigma_{\perp}^{2} + 3\left(\tau_{\perp}^{2} + \tau_{//}^{2}\right)} \qquad (1)$$

With:
σ_{eq} : Equivalent stress
$\sigma_{\perp}$: Normal stress orthogonal to weld throat
$\tau_{\perp}$: Shear stress orthogonal to weld axis
$\tau_{//}$: Shear stress parallel to weld axis
As shown in Figure 3:

For fatigue, analyses are performed at the weld toe and through the throat locations.

For weld toe fatigue assessment, [2] nodal forces have to be evaluated at the corresponding location. The weld leg edges are projected to the sheet mid-surfaces in order to force a node location. An imprint of those contours is performed as shown on Figure 4.

For weld throat fatigue assessment and according to [3], the weld section to be analyzed is in the leg plane, in the continuation of the root face.

The Fricke method calculates the maximum allowable stress which could be supported by a weld joint under fatigue loads. This approach is based on the linearization of the normal stress σ_x (z) within the weld leg section (see Figure 5). This stress is calculated from the nodal forces in this section. The two components of normal stress are evaluated: membrane stress and the bending stress, using the following formulas:

$$\sigma_{m,w} = \frac{1}{b.\lambda} \cdot \sum P_{xi} \qquad (2)$$

$$\sigma_{b,w} = \frac{6}{b.\lambda^{2}} \cdot \sum \left(P_{xi} \cdot \left(\frac{\lambda}{2} - z \right) \right) \qquad (3)$$

$$\sigma_{S,w} = \sigma_{m,w} + \sigma_{b,w} \qquad (4)$$

With:
$\sigma_{m,w}$: membrane stress
$\sigma_{b,w}$: bending stress
$\sigma_{s,w}$: structural stress
b: weld length
λ: leg length
P_{xi}: normal force

This method consists of calculating the resulting force and momentum at point B (Figure 6), which is located at the centre of the leg section. The position of the ideal weld shell element is conditioned by A and B point locations.

Using the shell element, the structural bending stress will be directly recovered using the node B momentum, with no integration through the thickness.

For 3D complex assembly, the load distribution depends on the components stiffness. In order to obtain the right load fluxes, the weld

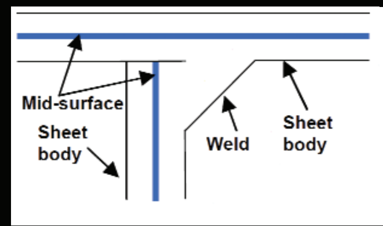


Figure 1: Sheet metal mid-surface

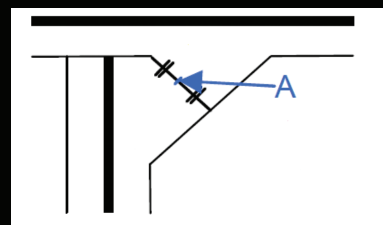


Figure 2: Weld throat section

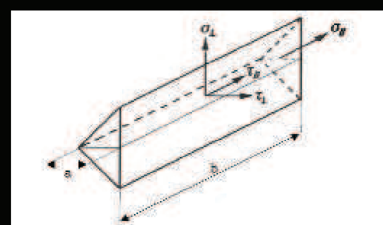


Figure 3: Weld stresses

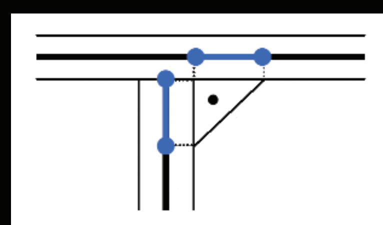


Figure 4: Weld leg imprints

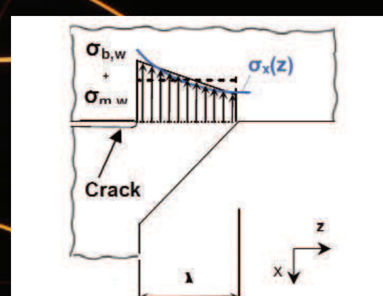


Figure 5: Stress linearization

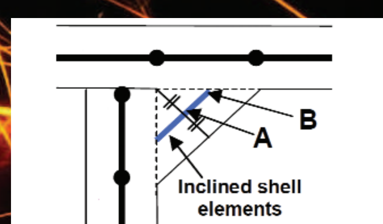


Figure 6: Weld modelling

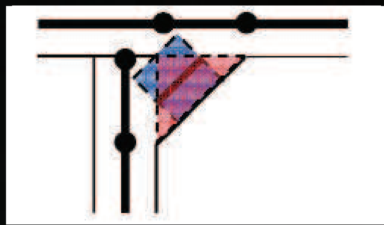


Figure 7: Equivalent section

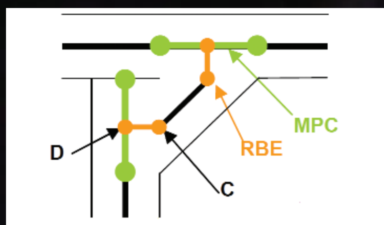


Figure 8: Connection

Figure 9a RBE Figure 9b MPC
Forbidden double dependencies

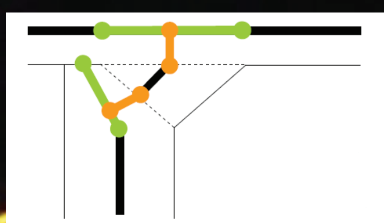


Figure 10: Fillet joint with penetration

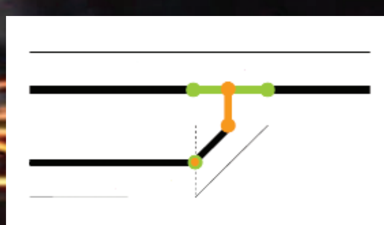


Figure 11: Overlap joint

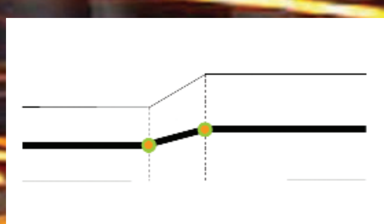


Figure 12: Groove joint

model stiffness has to be equivalent to the real stiffness of the seam weld joint. As illustrated in Figure 7, the LOHR weld shell element has the same section area as the weld and similar transverse and longitudinal stiffness. The blue section represents the shell element area and the red section represents the weld area.

The last step in the LOHR modelling method consists of connecting the weld shell element and the sheet solid mid-surfaces. This must be done with no modification of the joint stiffness. This connection is carried out using the gluing technique. It consists of a rigid body element (RBE) for node projection, and multi-point constraint (MPC) conditions over the associated sheet element.

Node C is projected on the associated face, on node D. The C and D nodes are linked with an RBE element which represents a rigid kinematic connection. The D node is linked to the associated element (on the imprint) with an MPC element. Displacements of the D node are mapped over all the nodes of the connected element, which is represented in green on Fig.8. The process does not generate any additional rigidity.

These specific elements are used regarding the following rules [4], where double dependencies are not allowed:

- for RBE, a slave group cannot have two master nodes (Figure 9a)
- for MPC, a slave node cannot have two master groups (Figure 9b)

The LOHR weld model can be extended to a partially penetrated fillet joint, overlap joint and groove joint as shown in Figures 10, 11 and 12.

Verification and Validation

Based on a steel tensile load specimen with six fillet welds, the shell model is compared to a fine solid model for verification. Validation is performed with physical tests. A tensile load of 25200±8400 N is applied.

Numerical comparison of LOHR's coarse shell mesh model with a fine solid mesh model is performed using SAMCEF solutions [5]. Due to symmetries, only one eighth of the geometry is meshed. The LOHR model is only composed of 163 nodes (1025 degrees of freedom) instead of 186 384 nodes (541 739 degrees of freedom) with the solid model.

Displacement is the first result that can be easily checked. Figures 16 and 17 show that the deflections are similar for both models. When we compare the maximum displacement for each model, we observe a 5.7% difference.

With regards to static strength assessment, Figure 18 shows the equivalent stress along the seam for the maximum test load, F=33 600 N. This equivalent stress varies from 224 up to 248 MPa. Hand calculation according to (1) gives an average value equal to 238 MPa. The shell weld modelling technique allows the equivalent stress results along the entire seam length to be obtained (Figure 18).

The next step consists of comparing the linearized structural stress approach using both LOHR's weld model and a solid model as described by Fricke. Membrane and bending stress ranges are evaluated from the internal nodal forces of the leg weld section elements.

Stress Range	$\sigma_{m,\nabla}$ MPa	$\sigma_{b,\nabla}$ MPa	$\sigma_{t,\nabla}$ MPa
shell	14,85	77,01	91,86
volume	14,85	76,86	91,71

We notice that the discrepancy between both approaches is less than 1%. In Figure 19, the structural stress distribution along the seam length is plotted. The coarse shell mesh results fit well with the fine solid mesh values. LOHR's method is appropriate for structural stress calculation.

Figure 20 shows the physical test specimen [6] and the resulting root cracking in the weld leg section. Test results give fatigue life for each of the seven tested specimens, and the fatigue life range band associated with the calculated structural stress is plotted in Figure 21.

The International Institute of Welding recommended criteria, the FAT80 S/N curve [3], used with the calculated stress range is conservative compared to the test data.

CAD-CAE Automation

LOHR designs large structures and uses NX from SIEMENS PLM Sofware as CAD and CAE pre/post solutions. The manual shell meshing of the weld connections is a very tedious job. Partial automation has been developed with C++ programs which use NX Open API commands and can be launched from the NX session using icons.

The steps used in the CAD-CAE process are illustrated in the following example, see Figure 22. It is representative of the connection complexity.

To prepare the meshing, the idea is to use the solid representation of the weld using NX weld assistant CAD tool. The selection of the two faces of the sheets to be joined provides the fillet weld seam location. The weld cross section size allocation allows the seam solid to be generated, see Figure 23.

After the CAD weld creation, the weld leg may be imprinted over the connected parts. A program makes it automatically, Figure 24.

Shell meshing will be performed on the idealized mid-surfaces. The mid-surfaces are generated using CAD functionality. LOHR has developed another program which checks the mid-surface generation and transfers the weld leg imprint to the idealized faces. In order to avoid complex scar boundaries only the envelope of the imprint is generated, Figure 25.

The solid seam is also idealized to a mid- surface using the CAD tool. All the connection information is reported to the idealized geometry using object attributes. When switching to the CAE NX Advanced Simulation application, the object attributes are transferred. The shell meshing is performed traditionally. Once meshes are ready, the edge-to-face or edge-to-edge connections are performed automatically, as in Figure 26. The program activates the NX 1D connections which use the previously described mesh connection technique (Figure 6).

The global construction is based on CAD and CAE features with a robust capability for updating. Figures 27 & 28 illustrate the CAE update after modification of CAD parameters. In the present example:

1. Thicknesses are decreased
2. The weld throat sizes are decreased
3. The plate is moved
4. The faces of the plates are moved

Back in the CAE environment, the update of the meshing is performed with no manual operation.

Summary

LOHR is a large welded-structure vehicle manufacturer who strives to provide integrated manufacturing process cost analysis as early as possible in the product design phase. In order to deal with seam weld-design optimization, the only issue is to accurately model the weld connection for the structural analysis. This innovative approach consists of a new weld seam representation using a shell element for global structural analysis. This special weld element enables to assess the weld throat strength; either for static load or fatigue load. For fatigue, the structural stress approach uses linearized stress distribution over the throat thickness which is well suited to shell element use. This method has been verified by comparing stress integration throughout the throat thickness using a fine solid fem model and a coarse shell model. The approach is validated, it correlates fatigue test results. The LOHR's weld shell element model must provide the correct assembly stiffness in order to have the appropriate load path behaviour. That is why the LOHR's weld model has the same cross-section area as the physical weld seam. The shell seam mesh is connected to the sheet solids using NX Advanced Simulation connectors which do not provide any additional rigidity in the assembly. Some automation toolboxes have been developed

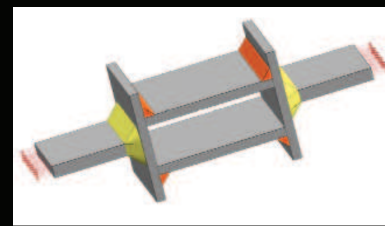


Figure 13: Tensile specimen load

Figure 14: Solid model meshing

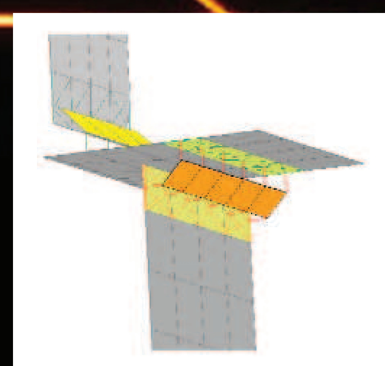


Figure 15: LOHR's shell model meshing

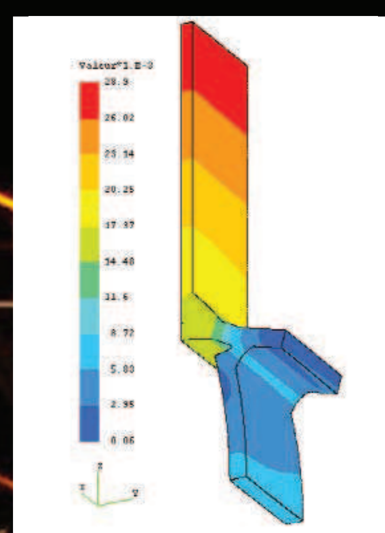


Figure 16:
Solid model displacement (mm)

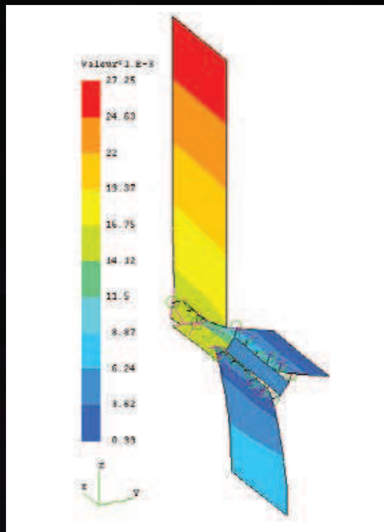


Figure 17: LOHR's shell model displacement (mm)

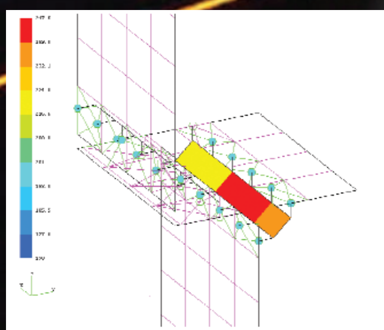


Figure 18: Equivalent static stress (MPa)

that help the model preparation process based on a CAD solid representation of the weld. Toolboxes first make the scar imprint of the solid seam over the welded plates. Then, when idealizing the geometry to mid-surfaces, the imprints are transferred to the mid-surfaces. The resulting scars may result in complex contours, mainly when the welding is made on both sides of a plate. Only the envelope boundary is retained for meshing simplification purposes. When shell meshing is performed, the last LOHR toolbox automatically makes the connection between the various meshes. This seam to simulation approach is very powerful when used in a "what if" variation context. Based on design parameter changes: plate thickness, weld size, face offset and solid motion., the robust capability of the CAD application allows the geometry to be updated. The mesh is also updated with no manual operation which allows easy additional design validations. The LOHR Industrie "seam sim" project provides a one-shot efficient global weld structural assessment tool in a unified CAD-CAE environment.

References

[1] Eurocode 3 / P 22-311-M "Design of steel structures" and National Application Document –Part 1-1: General rules and rules for buildings – Annex M: Alternative method for fillet welds / 1992

[2] SAE 982311 Mikael Fermér, Magnus Andréasson, and Björn Frodin, Volvo car corporation "Fatigue Life Prediction of MAG-Welded Thin-Sheet Structures"

[3] W. Fricke A. Kahl H. Paetzold «fatigue assessment of root cracking of fillet welds subject to throat bending using the structural stress approach», Doc IIW-1737-06

[4] SIEMENS PLM Software NX6 Advanced simulation help library.

[5] SAMTECH, Samcef User Manual V13.1

[6] Institut de Soudure RT-45058 : Fatigue test report

AUTHOR INFORMATION

D. Turlier, P Klein, A. Wilhelm and M. De Azevedo

LOHR industrie, France
Virtual.testing@lohr.fr

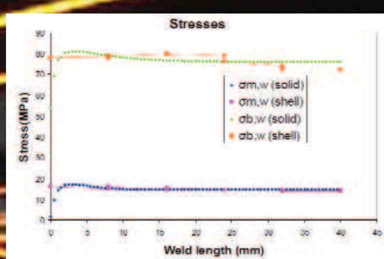


Figure 19: Structural stress range along the weld length (MPa)

Figure 20: Physical test - crack along the weld leg

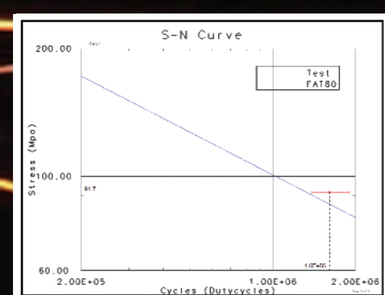


Figure 21: S-N curve

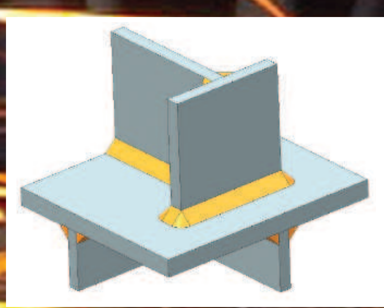


Figure 22: Weld-assembly specimen

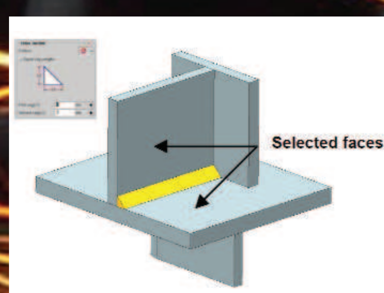


Figure 23: Weld solids

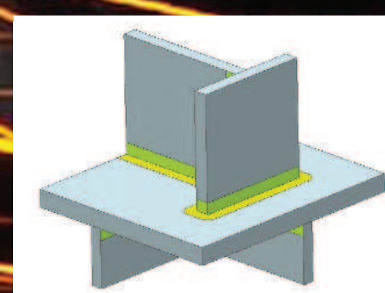


Figure 24: Weld leg imprint operation

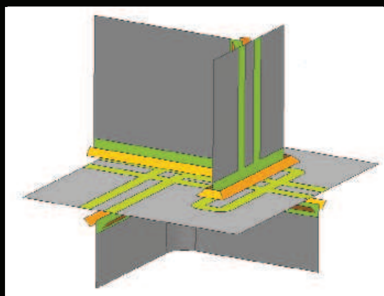


Figure 25: Imprint update to idealized geometry

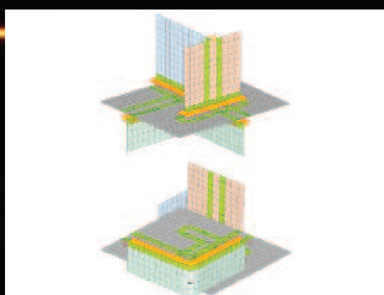


Figure 26: CAE connection operation

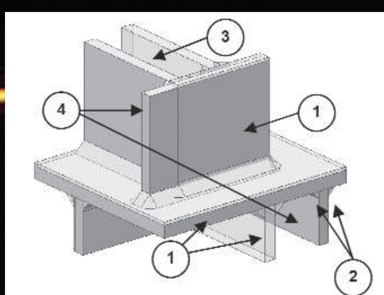


Figure 27: Geometry modifications

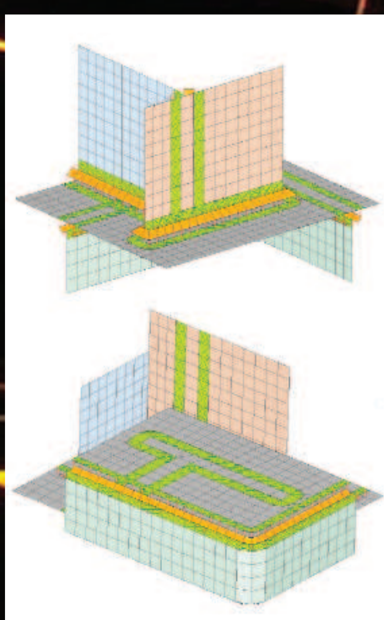


Figure 28: Update of the FEM