Društvo za tehniku
zavarivanja Slavonski Brod

12. Međunarodno znanstveno-stručno savjetovanje SBZ 2023

„STROJARSKE TEHNOLOGIJE U IZRADI ZAVARENIH KONSTRUKCIJA I PROIZVODA, SBZ 2023."
Slavonski Brod, 26. i 27. 04. 2023. i Požega 28. 04. 2023.

COMPARISON OF FILLET WELD CALCULATIONS USING SOFTWARES IDEA STATICA AND ABAQUS CAE

M.Vilić[1,*], J.Marijić[1], I.Grgić[1], M.Karakašić[1], Ž.Ivandić[1]

[1]Mechanical Engineering Faculty in Slavonski Brod, University of Slavonski Brod, Croatia

[*] Corresponding Author. E-mail: mvilic@unisb.hr

Abstract

Thanks to modern computer programs, a large number of calculations can be made easily and quickly. Welded joints are most commonly used in steel structures, and fillet welds are among the most commonly used types of welds. There are numerous calculation methods for fillet welds. The most commonly used methods are the classical analytical method according to standards such as EN 1993-1 (Eurocode 3) and the finite element method. This paper dealt with the calculation of fillet welds according to the finite element method. The calculations were performed using the computer programs IDEA StatiCa (IS) and Abaqus CAE (AS). IDEA StatiCa uses an adapted FEM analysis called CBFEM (Component Base Finite Element Method) by the developers of the program.

CBFEM is a type of FEM analysis where the user does not have much influence on the choice of the type of finite elements, the size of the finite element mesh, or the type of interactions between the finite elements. Most of the analysis is automated; both geometry and load cases can be imported from other programs. This type of calculation is very different from the classical FEM analysis, which requires the user's knowledge and experience to adjust all the parameters of the calculation. The classic FEM analysis requires more time and involves a much higher probability of error.

The aim of this work was to compare the differences between the classical FEM analysis of a welded joint and the analysis performed in IS, to compare the equivalent stress results, the time required for the analysis, the complexity of the setup and the level of knowledge required to use one tool and the other, and at the end to draw a conclusion on which type of analysis is better and why.

Keywords: Fillet weld calculation, IDEA StatiCa, Abaqus CAE, FEM analysis, steel structure

1. Introduction

Fillet welds are commonly used in structural steel connections to transfer loads between members. However, their analysis can be complex due to the nonlinear behavior of the weld material and the potential for various failure modes. To simplify this analysis, many programs, such as IDEA StatiCa and Abaqus, have developed methods to accurately model fillet welds.
IDEA StatiCa is a software tool designed to analyze fillet welds using the CBFEM method, which stands for Component Based Finite Element Method. This approach utilizes a nonlinear material model to accurately predict the behavior of the weld, taking into account effects of plastic deformation. With this software, engineers and designers can analyze the strength and deformation of the weld under different loading conditions. Additionally, IDEA StatiCa enables users to evaluate the fatigue life of the weld by considering various stress ranges [1].
On the other hand, Abaqus is a popular Finite Element Analysis (FEA) software program that can be used to simulate the behavior of fillet welds. It offers a range of material models that can accurately capture the nonlinear behavior of the weld. Abaqus also allows users to simulate the weld forming process to predict welding residual stresses and distortions, which can affect the performance of the weld [2].
Overall, using software programs such as IDEA StatiCa and Abaqus can provide accurate and detailed analysis of fillet welds. The results can be used to optimize weld designs, assess the structural integrity of existing connections, and ensure the safety of the overall structure.

2. IDEA Statica fillet weld calculation

2.1. Design phase in IDEA StatiCa

Design phase in software IDEA StatiCa is very user friendly and logical. User can be unexperienced in both FEM analysis and in steel structure design and still manage to setup analysis quickly and simply. In design phase user is prompted to choose class of connection, basic geometry, type of design and parameters as shown on Figure 1.

Figure 1. IDEA StatiCa basic setup

Parameters chosen are structural steel S355 according to standard EN 10025-2 and EN 1993 (EC3) for weld design. The connection consists of an IPE 220 (according to standard DIN 1025-5) connected to a HEA 200 (according to standard DIN 1025-3) via a double-sided fillet weld on both flanges and web. An encastre boundary condition is set on the HEA 200 ends, and a 70 kN force is applied to the free end of the IPE 220, as shown in Figure 2.

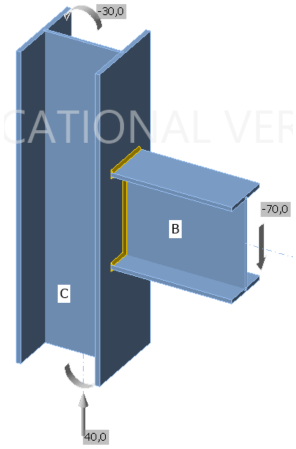
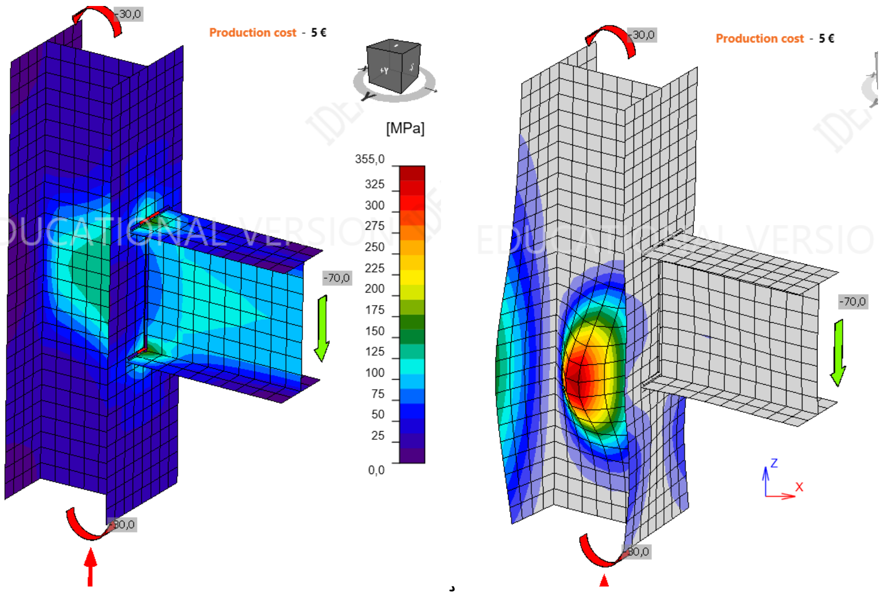


Figure 2. IDEA StatiCa basic setup

2.2. Results of analysis in IDEA StatiCa

IDEA StatiCa provided results rapidly. This particular calculation took only 3,5 minutes to setup and 3,2 seconds to finish calculation, which is astonishing. This kind of fast calculation allows user to try out many iterations on types of parameters. Results are provided according to chosen standard, EN 1993 in this case [3]. It provides results for equivalent stress in plates and welds, plastic strain and stress in contacts (for bolted connections) and shapes of buckling and eigenvalues. All of this is shown on Figure 3.



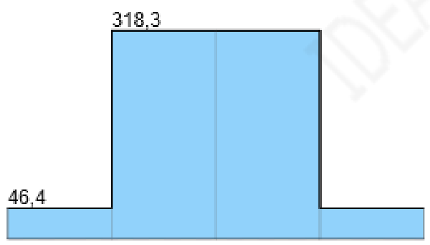
Local buckling of joint

	Loads	Shape	Factor
>	LE1	1	23,10
		2	28,32
		3	28,63
		4	36,72
		5	52,05
		6	52,80

a) b) c)

Figure 3. a) Equivalent stress distribution in plates and welds, b) First buckling mode of joint, c) Buckling modes and eigenvalues

Largest weld equivalent stress is 318,3 MPa and it is positioned at bottom flange of IPE 220 beam. Software also provides analytical calculations of all the welds based on EN 1993-1-8 and stress distributions. Examples of these are shown on Figure 4.



Weld resistance check (EN 1993-1-8 4.5.3.2)

$\sigma_{w,Rd} = f_u/(\beta_w \gamma_{M2}) =$ 435,6 MPa $\geq$ $\sigma_{w,Ed} = [\sigma_\perp^2 + 3(\tau_\perp^2 + \tau_\parallel^2)]^{0,5} =$ 318,3 MPa

$\sigma_{\perp,Rd} = 0,9\, f_u \,/\, \gamma_{M2} =$ 352,8 MPa $\geq$ $|\sigma_\perp| =$ 196,2 MPa

where:

$f_u = 490{,}0$ MPa – Ultimate strength

$\beta_w = 0{,}90$ – appropriate correlation factor taken from Table 4.1

$\gamma_{M2} = 1{,}25$ – Safety factor

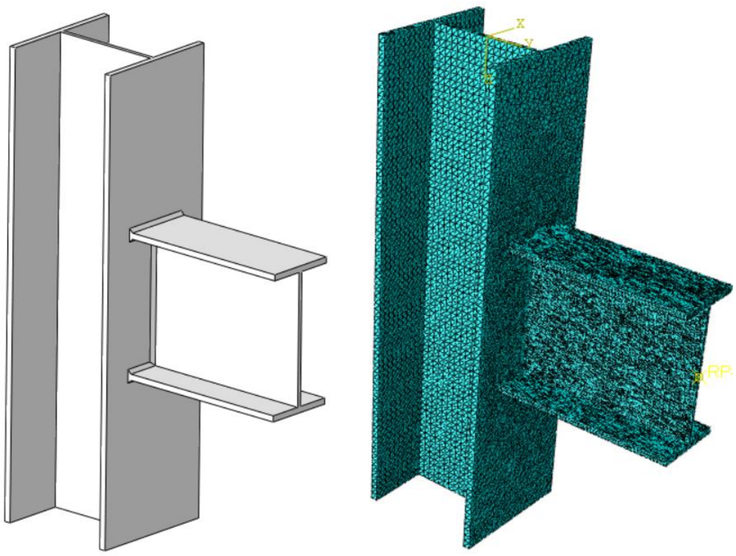
a) b)

Figure 4. a) Equivalent stress distribution in critical weld b) Analytical calculation of critical weld

3. Abaqus CAE fillet weld calculation

3.1. Design phase in Abaqus CAE

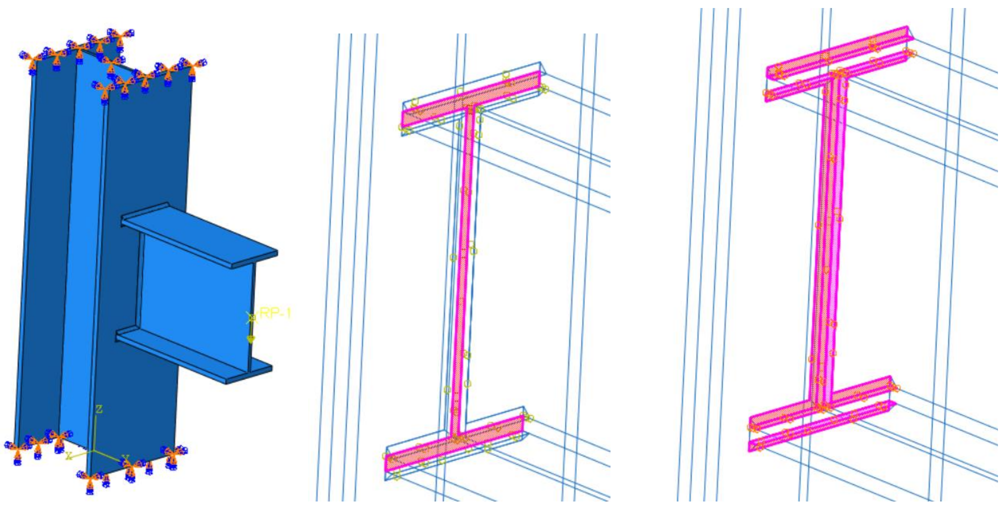
Design phase in Abaqus differs a great deal in comparison to IDEA StatiCa. It requires user to have more than basic knowledge in 3D modeling that is suitable for FEM analysis, material properties, boundary conditions, interaction properties, size and type of finite elements. Beams and welds require to be 3D modeled without any non-esential details so they are easily meshable and mesh quality will be much higher as shown on Figure 5.



a) b)

Figure 5. a) 3D model without too many details b) 3D model meshed with 10-node quadratic tetrahedron finite elements

In next steps user is required to setup materials properties (Young's Modulus and Poisson's Ratio), boundary conditions and loads, type of analysis and interaction properties. All of this requires experience and knowledge. Boundary conditions and interaction properties are shown on Figure 6.



a) b) c)

Figure 6. a) Encastre BC and concentrated force of 70kN, b) Surface to surface contact interaction, c) Tie constraint between weld and beam

3.2. Results of analysis in Abaqus CAE

Abaqus CAE provided results in 6 hours and 50 minutes which is much longer than IDEA StatiCa. Both calculations were done using same computer so computing resources were equal. Such a long period for gaining results creates issues in creating many iterations in calculation parameters. Results are not related to any kind of standard so it is up to user to interpret them correctly. Time to create whole setup was 4 hours that involved creating 3D model, applying material properties, making assembly, applying interaction properties, loads and BC's, meshing with finite elements. Largest equivalent stress value was 1013,7 MPa as shown on Figure 7.

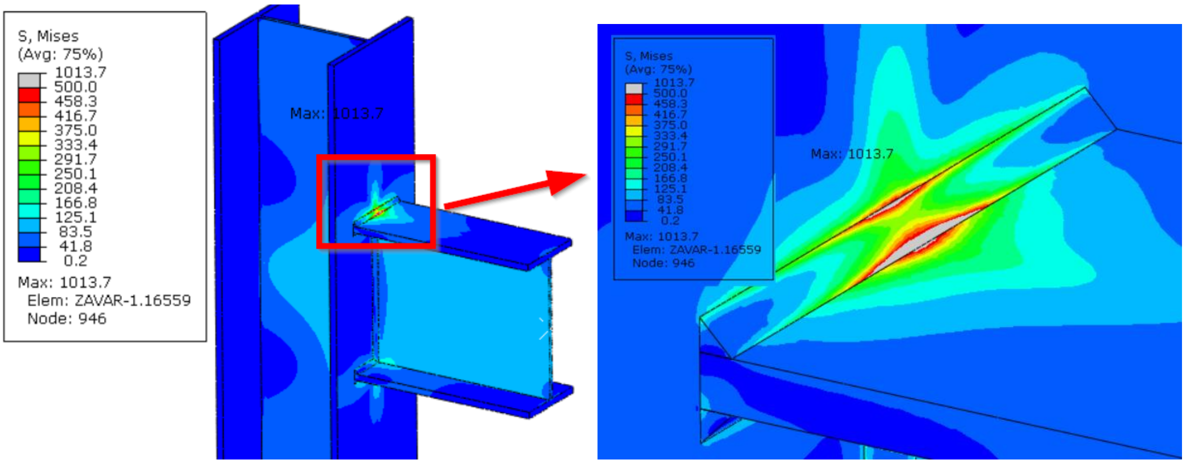


Figure 7. Equivalent stress distribution

4. Comparison of results

Both IDEA StatiCa and Abaqus use finite element analysis but provide very different results. IS used shell and AS 3D finite elements. IS interprets results automatically and correlates them with standard of choice (EC3 in this case) and AS leaves it up to user to interpret results. In table 1 below differences between two calculations are shown.

Table 1. Comparison of the two methods

	IDEA StatiCa	Abaqus CAE
Time to setup calculation	3,5 min	4 hours
Time to finish calculation	3,2 sec	6 hours 50 min
Largest equivalent stress value	318,3 MPa	1013,7 MPa
Complexity of setup	very easy	difficult
Level of knowledge necessary	Beginner	Experienced user

5. Conclusion

The results of our comparison between IDEA StatiCa and Abaqus CAE show that the time required to set up a calculation using IDEA StatiCa is significantly shorter, with an average of only 3.5 minutes, compared to Abaqus CAE, which took an average of 4 hours. The time required to complete a calculation was also much shorter using IDEA StatiCa, with an average of only 3.2 seconds, compared to 6 hours and 50 minutes using Abaqus CAE.

Furthermore, our analysis revealed that IDEA StatiCa is easier to use and requires less technical knowledge compared to Abaqus CAE, making it a more suitable option for beginners. The largest equivalent stress value obtained using IDEA StatiCa was 318.3 MPa, while Abaqus CAE produced a much higher value of 1013.7 MPa. Abaqus CAE is capable of producing more accurate and detailed results, especially in complex simulations, making it a very valuable tool for experienced users.

Based on our analysis, we conclude that IDEA StatiCa is the better software choice for this type of problem. Its user-friendly interface and efficient calculation time make it a suitable option for beginners and experts. Additionally, IDEA StatiCa automatically correlates its results with standards, such as Eurocode 3, providing a more straightforward and reliable interpretation of the results. IDEA StatiCa provides an optimal solution for engineers and designers looking for an accessible and efficient tool.

6. References

[1] https://www.ideastatica.com/ (accessed 14.04.2023)

[2] https://www.3ds.com/products-services/simulia/products/abaqus/abaquscae/ (accessed 14.04.2023)

[3] EN 1993-1-1 (2005) (English): Eurocode 3: Design of steel structures - Part 1-1: General rules and rules for buildings [Authority: The European Union Per Regulation 305/2011, Directive 98/34/EC, Directive 2004/18/EC]