buildings

MDPI

Article

Investigation of Stress Concentration Zones in FEM-Based Design of Welded Plated Structures

Erzsébet Bärnkopf, Balázs Kövesdi * and László Dunai

Department of Structural Engineering, Faculty of Civil Engineering, Budapest University of Technology and Economics, Műegyetem rkp. 3., H-1111 Budapest, Hungary; barnkopf.erzsebet@edu.bme.hu (E.B.); dunai.laszlo@emk.bme.hu (L.D.)

* Correspondence: kovesdi.balazs@emk.bme.hu; Tel.: +36-1-463-1726; Fax: +36-1-463-1784

Abstract: The numerical model-based design is commonly applied to steel structures using advanced numerical models and analysis. These models often contain stress concentration zones, which can cause problems for designers within the evaluation process. There are two basic questions to answer in the design: (i) are these stress concentrations real physical stresses or numerical singularities and (ii) should these stresses be considered in the design process or can be neglected? The current paper shows a proposal to separate the real physical stresses from the numerical stress concentrations and an improved design method is introduced to consider or neglect them in the daily design. The proposed evaluation method is presented through a design example taken from the daily bridge design practice. The calculation method of the design check is presented first by using (i) linear analysis and (ii) geometrical and material non-linear analysis. Based on the comparison of these two calculation methods the evaluation process of the stress concentration zone is presented as an example. The paper introduces an evaluation method for the stress concentration zone, which can be applied to different structures similarly.

Keywords: stress concentration; FEM-based design; numerical singularities; plated structures

Citation: Bärnkopf, E.; Kövesdi, B.; Dunai, L. Investigation of Stress Concentration Zones in FEM-Based Design of Welded Plated Structures. *Buildings* **2023**, *13*, 1057. https://doi.org/10.3390/buildings13041057

Academic Editors: John Papangelis and Muxuan Tao

Received: 18 March 2023
Revised: 4 April 2023
Accepted: 13 April 2023
Published: 17 April 2023

1. Introduction

The numerical model-based design of steel structures gives a new achievement to the designers to create new, innovative, and economic steel structures. These advanced numerical models often contain stress concentration zones at the load introduction locations and intersection points of different plate elements. The evaluation of the computed stresses can be challenging for designers. Traditional design methods are developed for "nominal stress theory" linked to beam-type numerical models, which do not contain stress concentrations. If advanced numerical models are applied, stress concentrations are included in the numerical calculations, therefore, design theories should be revised and improved. The aim of the current research program is to develop a general methodology for the consideration and investigation of stress concentration zones applicable in the design practice.

It is known that consideration of the stress concentrations can be different for different limit states: (i) elastic limit state—stress check, (ii) plastic limit state—progressive plastic flow, (iii) stability limit state, and (iv) ductile fracture limit state—tearing of the material. The focus of the current paper is put on the elastic limit state check using advanced numerical models. Within the current research program, the effect of the stress concentration zone is investigated, and which correct consideration is still not solved in the international literature, except for fatigue assessment.

Within the current research program, a general method for the consideration and evaluation of the stress concentration zones is presented, which is in line with the new Eurocode standard (prEN 1993-1-14 [1]), currently under development. The current paper gives explanations and background information on the new code. It also gives extended proposal for the evaluation strategy of the stress concentration zones, which follows the general Eurocode design rules, but it gives a more exact solution for design purposes. The application of the proposed design strategy is presented on a structural detail taken from the bridge design practice. A numerical model of the analyzed structural detail is developed, on which linear and non-linear analyses are executed to determine the nominal stresses, elastic stress concentrations, and plastic zones. Resistance of the structure is also determined using geometrical and material non-linear analysis. The size and development of plastic zones are investigated and the effect of the finite element shape and size on the numerical results is also evaluated. Conclusions are drawn regarding the consideration of the physical stress concentration within the linear analysis and the plastic strain evaluation within the non-linear analysis. The presented evaluation method can be applied to different structures in a similar way and can give a basis for a general design method development.

2. Literature Review

2.1. Previous Research Activities

There are relatively few studies focusing on the effect of the stress concentration zone or the effect of finite element type selection on the design check of welded plated structures. The physical stress concentration has been widely investigated, especially in the field of mechanical engineering and design recommendations applicable to the numerical model-based design are developed. An overview of these recommendations and design experiences can be found in [2]. It can be also seen, there are several specific investigations focusing on the accurate modelling of steel structures and investigating the effect of the FE mesh size on the calculated stresses and strains, but these investigations are mainly focusing on one specific problem and do not give general solutions. Such investigations can be found in [3–11], and a summary of them is published in [12].

Regarding the numerical model-based design recommendations, Oldal [2] investigated two connected plates using a 2D model and studied the effect of element size on the results for linear and second-order analysis. He concluded that using the second-order approximation function gives a more accurate result. Oldal did find it dangerous that the user-friendly interface of modern finite element software is easy-to-use so that a deeper, comprehensive knowledge of the finite element method is not required, leading to mistakes, and equally problematic if results are not evaluated in a way considering the limitations of the chosen model. He highlighted the paramount importance of correctly judging what is the right element size to obtain a sufficiently accurate solution to the problem, which leads to the basic dilemma of accuracy and efficiency.

Nemade and Shikalgar [3] demonstrated the mesh size effect; they modelled a 3 mm thick steel plate with four bolt holes and a welded rectangular tube connected to it. The plate was fixed and supported through four bolts, and the pure tensile load was placed centrally at the center of the tube. Four different meshing strategies were used in their finite element models (called: low-density free mesh, high-density free mesh, low-aspect-ratio structured mesh, and high-aspect-ratio structured mesh). The equivalent stresses as well as the total strains were investigated. It was recognized that the highest stress is generated in the region near the cylindrical edge of the holes. They concluded that the mesh density should be high enough in the area of stress concentration to accurately determine the real stresses and strains. The accuracy of the results is significantly dependent not on the type of meshing but primarily on the mesh density.

More and Bindu [8] investigated the effect of mesh size on the accuracy of numerical analysis. Static and buckling analyses were performed on the model of a steel-plated structure. Within the analysis quadratic shell elements and an automatic meshing technique were applied. Static analysis was performed first, and the von Mises stresses and strains were compared. It was assumed that the FE model with the finest mesh generated the most accurate results, and the percentage of approximate errors were calculated by comparing the other results with the most accurate one. They found that the errors of deformation were smaller than the errors of the von Mises stresses. It was also noticed, there are large differences in the stress concentrations by using the different models. It was emphasized that the computation time increases significantly with decreasing mesh size since the computation time of the coarse mesh model was less than 1/31st of the time used by the finest model. It was also determined which FE model is the optimal choice for static analysis in terms of accuracy and efficiency depending on the applied mesh size.

Based on the literature review, it can be concluded that all the previous investigations were focusing on the modelling of specific problems and evaluated the results using different mesh sizes or element types. However, there are no general design recommendations covering a larger part of the steel structural design.

It is also important to highlight that errors in the numerical model can have different sources, e.g., approximation errors, errors in the choice of grid size, errors in the computational algorithm, and rounding errors in the result. In the current paper, a methodology will be introduced to separate the total error containing all the possible errors from the real physical stresses (which are not errors, but stress peaks in the numerical calculations). The current research focuses (i) on the separation of the numerical errors from the real physical stresses and (ii) on the consideration of the physical stress concentration in the static check. However, it does not check the reason for the numerical errors.

2.2. Provisions of prEN 1993-1-14 and Its Background

The new Eurocode standard (prEN 1993-1-14: Design assisted by finite element analysis; scheduled publication date is 2024) has the aim—among others—to fill the above-mentioned gap. It has the task to give advanced design rules for numerical model-based design. One of the most important points of this code is introducing and separating different design methodologies, which are applied to steel structures and giving application rules for these methodologies. The code prEN 1993-1-14 in Section 8.1.2 contains design rules for the consideration of stress concentration zones as well, which can have two different sources:

- Physical stress concentration,
- Numerical singularities.

These two phenomena usually exist in combination in the numerical model and the first task of the designer is their separation. It is an important point because the consideration of the stress concentration is different for the numerical singularities and the physical stress concentrations. For the separation, there are currently two different approaches proposed by the prEN 1993-1-14 Annex B1:

- Calculation of mesh-independent stresses or strains, and
- Implementation of a circular rounding at the location of the sharp edges/corners.

If sharp edges/corners are used in the numerical model the numerical singularities cannot be fully avoided and they cannot be entirely separated from the physical stress concentrations. In the last corner elements, the numerical singularity always plays a role. In the case of models having sharp corners one possible approach to separate the geometrical (physical) stress concentration and the numerical singularities is based on the determination of the mesh-independent stresses or strains. The mesh-independent stresses or strains are calculated values at integration points of the elements which are not affected by further FE mesh refinement, as shown in Figure 1. The horizontal axis of the graph shows the position along the evaluation line (shown by the bold red line on the numerical model on the right-hand side). The vertical axis presents the calculated normal stresses. The figure presents numerical results belonging to 6 numerical models using different mesh sizes. The comparison of the calculated stresses showed that there are data points, where the computed stresses fit each other (within max. 1% difference) and in the corner element, at the maximum point the difference can be large. Those data points, where all calculations result in (except for the last finite element) equal stresses, are called mesh-independent stresses. The data points outside of this region are called numerical singularities. Using this evaluation methodology, the following terms can be clearly separated: (i) nominal stresses, (ii) physical stress concentration, and (iii) numerical singularities.

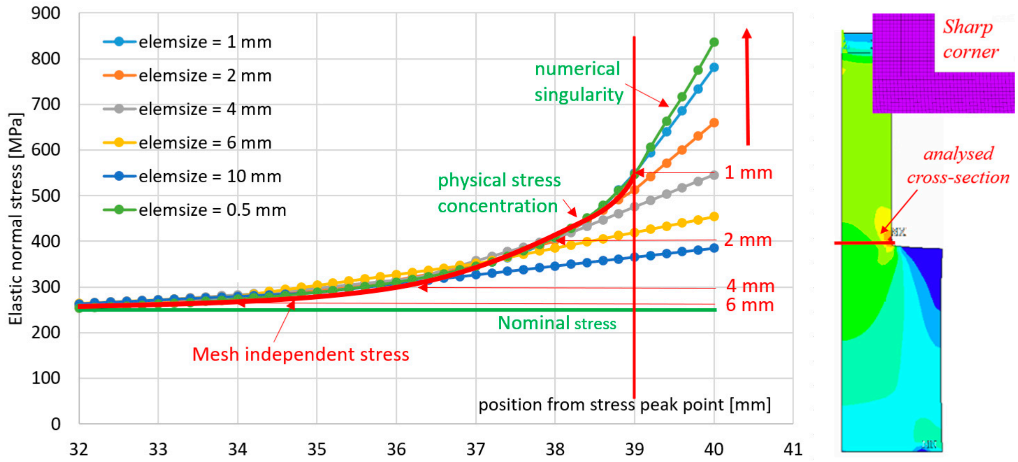

Figure 1. Determination of mesh-independent stresses [13].

Based on the results, it is also observed that by mesh refinement an increased part of the geometric stress concentration can be estimated, and the zone of the numerical singularities can be reduced. The mesh-independent stresses can only be defined in the zones where calculation results using different FE mesh sizes are existing and the calculated values are identical (e.g., differences are smaller than 1% for all applied mesh sizes).

Another optional way is to implement a circular rounding in the geometry at the location of the sharp edges/corners to avoid numerical singularities and to make the stress distribution smoother. One example is shown in Figure 2. If circular rounding is applied in the numerical model, by using a smooth and regular mesh, the numerical singularity can be avoided, and the analytically calculated stress concentration can be achieved with high accuracy. For one example, the comparison of the analytically and numerically calculated values is shown in Figure 2 taken from [12], where the stress concentration value depends on the D1, D2 values, and rounding radius (r).

In this case, the size of the rounding radius and the mesh size has a significant impact on the value of the geometrical stress concentration. Therefore, special attention should be given to their values, where engineering judgement or real values can be used. At those locations where the geometry of the structure has also a circular rounding, the rounding radius used in the numerical model should be identical to the real values. At those locations, where the geometry of the structure has no circular rounding, only welds as applied (for example), the rounding radius to be used cannot be clearly defined. Ongoing research activities are running now to find the appropriate values for various applications and failure modes. Currently, there are no general recommendations, except for the fatigue design situation.

The new code says in Section 8.1.2(4), the numerical stress concentrations (singularities) may be neglected in the design as they result from errors of the numerical approximation of the physical stresses or strains. However, Section 8.1.2(5) says the geometrical stress peaks should be considered or neglected in the design depending on the chosen analysis method and limit state criteria. It means the need to consider stress concentration depends on the limit state criteria to be checked. The normative part of the code does not give an exact evaluation strategy for the designer. However, an informative Annex (EN 1993-1-14 Annex B) gives guidance on the consideration of the stress concentration zones, the normative standardization requires further research. The subject of the current paper attempts to fill this gap. The informative Annex says, that for numerical design calculations using analysis requiring subsequent design check, stress concentration can be neglected in the following cases: (i) elastic stress check, (ii) determining of stresses or internal forces used in elastic or plastic strength check, or (iii) stability checks. It means, extrapolated nominal stresses can be used for the evaluation of the utilization ratio. For the calculation of the extrapolated stresses the mesh-independent stresses can be used, as shown in Figure 1. The reason why the effect of the stress concentration can be neglected is that in the case of numerical design calculations using analysis requiring subsequent design checks, the numerical model is used to determine the internal forces and deformations, which are further evaluated and compared to the limit values according to the applied limit state criteria. These criteria are determined for hand calculation needs, where the stress concentrations are not considered. Therefore, to ensure the harmonization of the model level and limit state criteria, the effect of stress concentrations can be neglected.

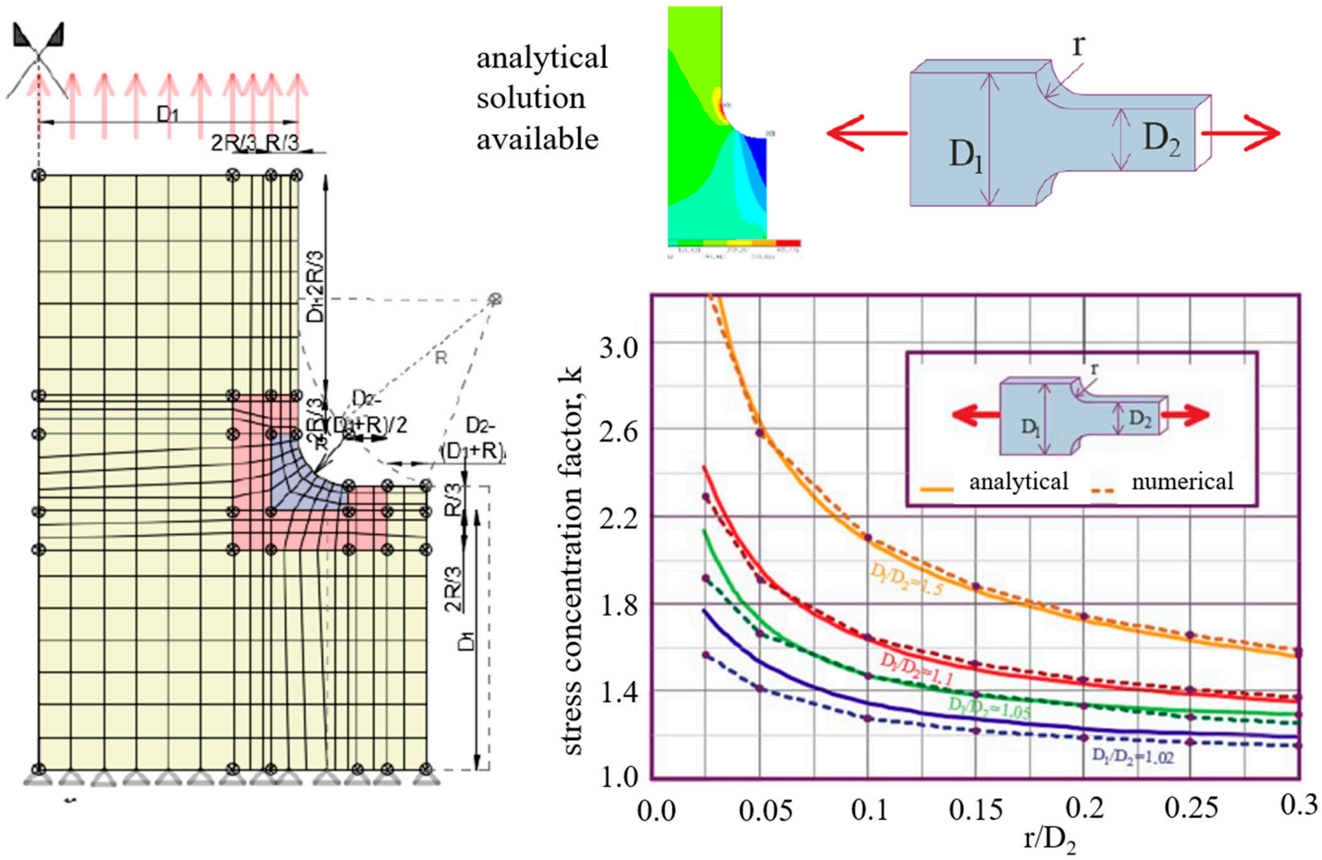

Figure 2. Comparison of analytically and numerically calculated stress concentration [12].

However, stress concentration should be not neglected if fatigue or fracture limit states are checked, where the limit values are harmonized with the existence of the stress concentrations (fatigue limit state check) and plastic elongations (plastic fracture limit state check, e.g., tension member net section failure, bolted joints, welded joints, etc.). In the case of numerical design calculations using direct resistance check, the effect of stress concentration is implicitly covered by the numerical model and the applied failure criteria (maximum plastic elongations should be also checked).

However, the code says if geometrical stress concentrations are neglected, and the need for the check arises, the stress concentration may be checked by an additional material non-linear analysis, limiting the maximum plastic strains within the stress concentration zone. In this approach, mesh-independent plastic strains should be determined similarly to mesh-independent stresses. Therefore, the material non-linear analysis should be executed to follow the distribution of the plastic strains within the stress concentration zone. For the maximum stresses, it can be also observed, by mesh refinement the maximum plastic elongations are increasing if sharp corners are applied. If rounded corners are used, the maximum strains are increasing by reaching a maximum value that depends on the applied rounding radius, as shown in the example of Figure 3. The diagram shows, if sharp corners are applied and the FE mesh size is decreased, the maximum elongations at the corner point are increasing and going to singularities. Singularity can be avoided by modelling rounded corners. The maximum plastic elongations, however, are strongly dependent on the applied rounding radius. There is one common property within the calculated values, using mesh refinement all diagrams are tending to a maximum value and not going to infinity. It means, the maximum plastic elongations can be determined, and it should be compared to the maximum allowed plastic strains. The maximum value depends on the rounding radius, which has special importance in the calculation process. To the accurate application of this calculation method and to the determination of the rounding radius, there are currently no general design rules.

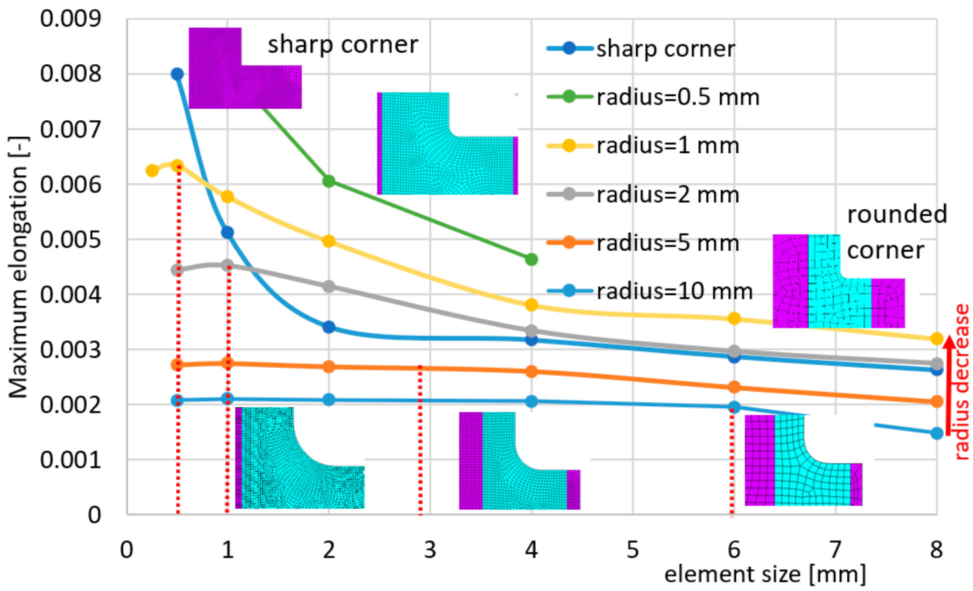

Figure 3. Effect of rounding radius and FE mesh size on the maximum plastic elongations.

The current paper aims to demonstrate the correct application of the above-described design rules, and it gives an extension of the rules by a general strategy, which can be followed in the design practice on the evaluation of the stress concentration zones.

3. Proposed Evaluation Strategy

To prove the accuracy of the above-presented method and its applicability in daily design, one typical structural detail of a steel bridge having significant stress concentration is presented in the following section. The presented example has two aims: (i) presenting the correct evaluation process of the stress concentration zone, and (ii) proving the applicability of the design rule, saying that the effect of the stress concentration can be neglected, and the extrapolated nominal stresses can be used for design check. The investigation strategy applied can be summarized, as follows:

(i) Evaluation of the stress concentration zones and determination of utilization ratio:

1. Advanced numerical model is developed using shell elements.
2. Loads corresponding to the design load level (called load level 01) are applied to the numerical model.

Level I analysis

3. Linear analyses are performed by using different FE mesh sizes; the effect of mesh size on the stress concentration zone is evaluated at the design load level.
4. Effect of the applied finite element type used is also investigated and the differences in the stress concentration zones are determined.
5. Mesh-independent stresses are determined and evaluated on a graph depending on the applied mesh size.

6. The mesh-independent stresses are extrapolated leading to the maximum stresses of $\sigma_{nom,max}$.

Level II analysis

7. If the results of the linear analyses show the maximum stress (σ_{max}) is larger in the stress concentration zone than the yield strength of the applied steel material, further investigations should be carried out to investigate the maximum strains within the stress concentration zone using non-linear analysis and an elastic–plastic material model. The distribution of stresses and strains is investigated at load level 01.

(ii) Proving the accuracy of the EN 1993-1-14 design rule:

8. Geometrical and material non-linear analysis is performed to determine the load-carrying capacity of the analyzed detail. The load–deformation path of the structure is evaluated. The ultimate load and the end of the linear behaviour are both determined. The end of the linear behaviour is calculated based on a comparison of GMNA (geometric and material non-linear analysis) and GNA (geometric non-linear analysis) analysis.
9. Based on the comparison of the non-linear and linear analysis results, the stresses to be considered within the linear analysis are back-calculated and design recommendations are developed for the consideration of the mesh-independent stresses.
10. Evaluation methodology for the plastic strains is also introduced for the investigated structural detail; the utilization ratio based on the plastic strains are also determined.

The design flowchart representing points 1–7 are shown in Figure 4. Both LA and GMNA analyses are carried out at the design load level.

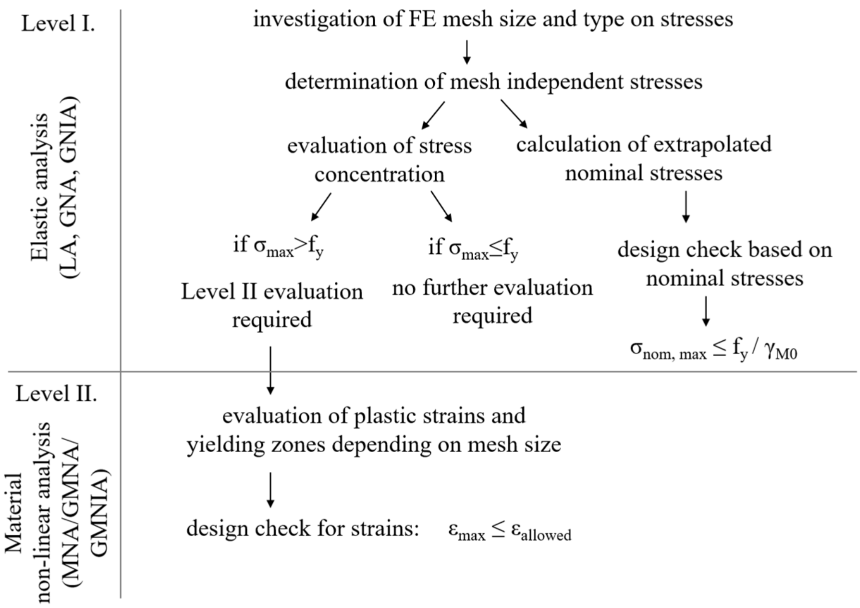

Figure 4. Strategy of the evaluation process for elastic limit state check.

Points 8–10 are additional investigations proving for the analyzed particular structural detail the Level II analysis—non-linear calculation—could be supplemented by a simple stress check using extrapolated mesh-independent nominal stresses based on the LA calculation. The same evaluation strategy could be further used on different structural details which could finally lead to the categorization of structural details from the stress concentration point of view, similar to the fatigue design of steel structures.

It is also important to mention that the above-described strategy is not applicable to fatigue assessment, which has its own rules by applying (i) a hot-spot stress approach or (ii) a notch stress approach. The aim of the current investigation is to create clear design rules to be used within the elastic or plastic check of steel structures, which could solve the problem of the stress concentration similarly to how the hot-spot or the notch stresses solved it for fatigue problems.

4. Case Study: Welded Stiffener-to-Cross-Girder Connection with Cut-Out Holes

4.1. Numerical Model Development

The analyzed detail is a typical steel bridge with an orthotropic deck system. Closed-section trapezoidal longitudinal stiffeners and I-shaped cross-girders are supporting the deck plate, as shown in Figure 5. The model includes two times two meters of the structure in both directions from the cross-girder. The main dimensions used in the modelling are given in Table 1.

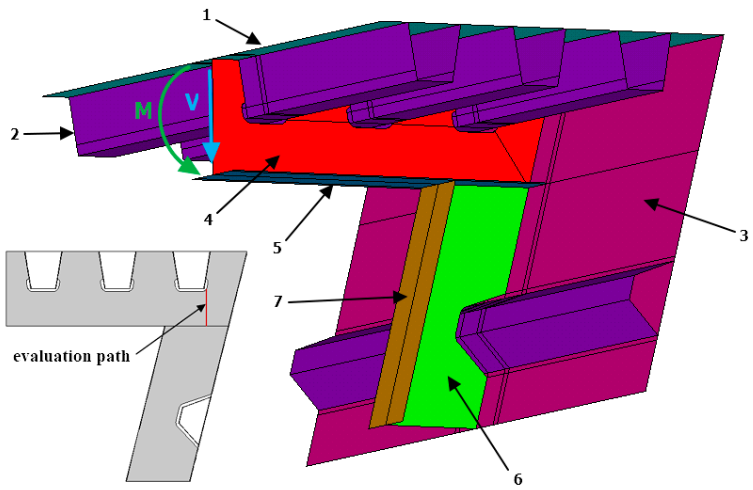

Figure 5. The modelled detail, marked with each structural element for identification, the loading conditions, and location of the evaluation path in the model.

Table 1. Dimensions of the model components.

	Element Number	Dimensions (mm)
deck plate thickness	1	14
longitudinal stiffener thickness	2	8
web plate thickness	3	12
cross-girder web height	4	600
cross-girder web thickness		12
cross-girder flange width	5	250
cross-girder flange thickness		16
web transversal stiffener web height	6	500
web transversal stiffener web thickness		16
web transversal stiffener flange width	7	200
web transversal stiffener flange thickness		16

Each element in the table is assigned a number, which is used to identify the structural elements. Figure 5 also shows the location of the evaluation path where the stress concentration zone is studied, and the mesh-independent stresses are determined.

The model is supported at both ends against displacement in all directions, as presented in Figure 6, showing the applied finite element mesh, boundary, and loading conditions. The lower edges of the transverse stiffener of the web are also supported in this way. The detail is loaded in shear and bending moments. The shear force is distributed at the edge nodes of the cross-girder web, directed downwards, as a uniformly distributed load. The bending moment is applied as a force couple at the flanges of the cross-girder, also as a uniformly distributed load. The bending moments cause tensile stresses at the edge nodes of the deck plate and compression at the bottom flange of the cross-girder.

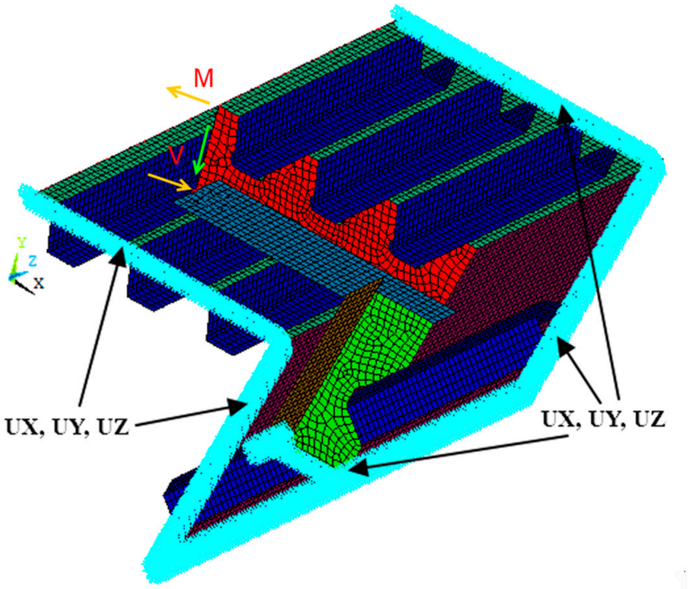

Figure 6. Finite element model (mesh, boundary, and loading conditions).

The numerical model development is performed using Ansys 19.2 [14] finite element software. The study investigates the effect of different finite element types and mesh sizes. Twelve different mesh densities are used in the model, using the maximum geometrical measures as 200, 150, 100, 75, 50, 40, 30, 25, 20, 15, 10, and 5 mm. The structural details are analyzed using (i) eight-node (SHELL281) quadrilateral-shaped, (ii) eight-node triangle-shaped, (iii) four-node (SHELL181) quadrilateral-shaped, and (iv) four-node triangle-shaped thin shell elements. The analyzed structural detail is welded stiffener-to-cross-girder connection with cut-out holes. This structural detail is loaded by the combination of bending moment and shear force, therefore, the von Mises stresses are evaluated within the stress concentration zone.

4.2. Linear Analysis—Evaluation of Stress Concentration

In the linear analysis, internal forces of M = 360 kNm and V = 72 kN are applied. These loads are treated as design load levels (load level 01), for which the elastic limit check should be performed. The von Mises stress distribution acting in the cross-girder web is shown in Figure 7. Results show stress concentration develops around the cut-out of the longitudinal stiffener on the cross-girder web.

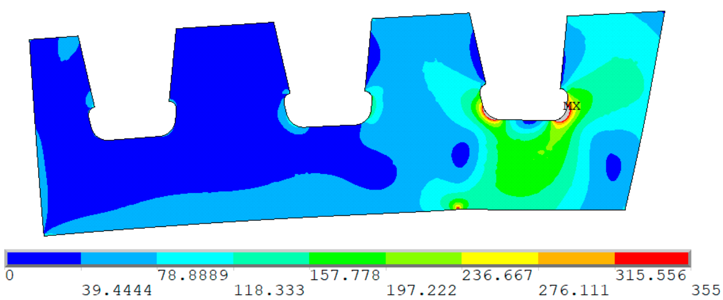

Figure 7. von Mises stress distribution on the cross-girder scaled to the yield strength (units in (MPa)).

The effect of the finite element type and shape on the stress distribution is investigated first. The maximum stresses obtained in the four cases are plotted as a function of mesh size shown in Figure 8. The maximum values of the calculated stresses are presented here for each applied FE mesh, which are taken from the evaluation path shown in Figure 9. The horizontal axis shows the applied mesh size, and the vertical axis presents the maximum obtained stresses in the peak point of the stress concentration zone. Results show for a sufficiently dense mesh there is no significant difference between the results obtained using different element types. The most accurate results are obtained by the quadratic shell elements (SHELL281) having a triangle shape. In the case of linear shell elements (SHELL181) a much finer mesh is necessary to obtain the same accuracy as using coarse meshes with quadratic elements. There is also a significant difference between the triangle and quadrilateral element shapes. In the analyzed case mesh with triangle-shaped elements gave more accurate results. Therefore, quadratic shell elements with a triangle shape are applied within the further analyses.

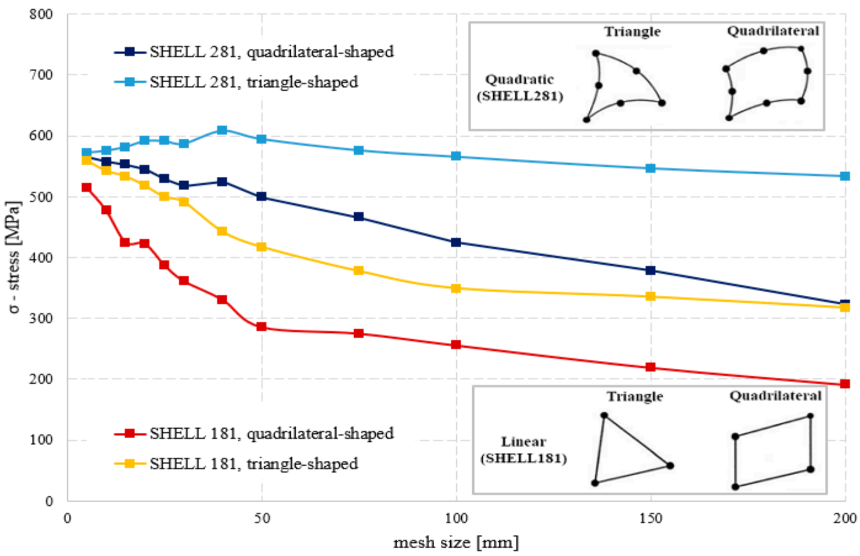

Figure 8. Effect of finite element type and shape on the maximum stresses.

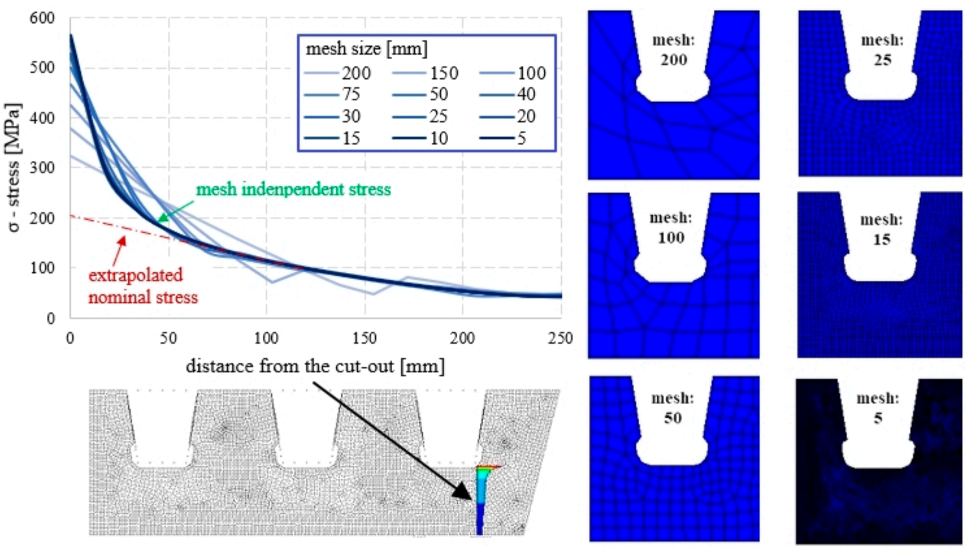

Figure 9. von Mises stresses obtained from the linear analyses along the evaluation path using different mesh sizes (units in (MPa)).

The effect of the mesh size on the stress concentration is also analyzed. The magnitude of the von Mises stresses is calculated for each analyzed mesh size along the evaluation path. Results are presented in Figure 9.

The computed stresses are plotted as a function of distance from the cut-out. The large variation of the stress peak points can be observed on the diagram. The maximum stress significantly increases by decreasing the mesh size and the extension of the stress concentration decreases, which is typical for stress concentration zones. The bold curve in Figure 9 represents the mesh-independent stresses as a result of the mesh sensitivity study. The red dashed line represents the extrapolated nominal stresses, which has a maximum value of 200 MPa at the location of the stress peak point. The extrapolated nominal stresses can be used for the resistance check and for the calculation of the utilization ratio within the elastic limit state check. It means, using S355 steel grade, this stress level corresponds to a utilization ratio of 56% applying partial factor γ_{M0} = 1.0.

The second point to be checked is the maximum value of the mesh-independent stresses, which is far above the yield strength (f_y = 355 MPa—limit function $\sigma_{max} > f_y$ according to the workflow given in Figure 4). The mesh-independent stress represents the physically correct stress distribution of the analyzed structural detail. This result means, at the analyzed design load level, the vicinity of the stress concentration zone yields. Therefore, the maximum strains should be checked within the verification process by applying material non-linear analysis.

Geometrical and material non-linear analyses (GNMA) are performed, and the non-linear behavior of the analyzed structural detail is evaluated. For this case no instability problem can occur at the stress concentration zone, therefore, no imperfections are applied to the numerical model. The application of a material non-linear analysis (MNA) would result in similar strains to a GNMA. In the analysis, S355 steel grade is used with a yield strength of f_y = 355 MPa and elastic modulus of E = 210 GPa. The stress–strain diagram used is shown in Figure 10. Linear elastic—hardening plastic material model using the von Mises yield criterion is applied. After reaching the yield strength, the material model has a hardening behavior with a minimum increase necessary to avoid numerical errors. Significant hardening of the material is neglected within the analyses to keep the results of the numerical calculation compatible with the hand-calculation-based linear analysis and to avoid overestimation of the resistance which could come from the effect of strain hardening. The applied material model also contains a damage criterion to limit the maximum allowed plastic strains the material can perform. By reaching the maximum allowed plastic strains the material model would identify the loss of strength of the elements modelling the damage to the material. The failure limit point is defined by the strain level which belongs to the tensile stress of the material. The strain level at the ultimate tensile stress was taken as ~10%. To describe the progressive crushing of a material the damage function is used to reduce the material's yield strength. If a state of stress is found to lay outside of the yield surface a backward-Euler algorithm is used to return the stress to the failure surface. The resulting inelastic increment in strain is then accumulated as a crack strain. The maximum stress that can be sustained in an element is then reduced as a function of crack strain according to the material model description of Ansys [14]. Using this material model the softening slope should be defined by the user. The effect of the softening slope (shown in Figure 10) is investigated, and it is found that in the current case, it has no effect on the resistance of the analyzed structure.

Based on the non-linear analysis, the strains within the stress concentration zone are determined and their maximum value is evaluated. The computed strains along the evaluation path are presented in Figure 11a for one specific mesh size. As the maximum value is dependent on the mesh size, the peak strains are evaluated by using different mesh sizes; results are shown in Figure 11b. Results prove, by decreasing the mesh size, the maximum strains are increasing but tending to a maximum value, which is 0.29% in the presented case.

The results show the maximum value of the calculated strains at the stress peak point (0.29%) is significantly smaller than the allowed plastic elongation of the steel material (physical limit value is around 20% for S355 steel; prEN 1993-1-5 gives 5% as a maximum allowed plastic strain). It means the utilization ratio regarding the strains would be 5.8%, which is significantly smaller than the calculated utilization ratio according to the extrapolated nominal stresses. These results indicate the stress check is decisive for the analysed structural detail.

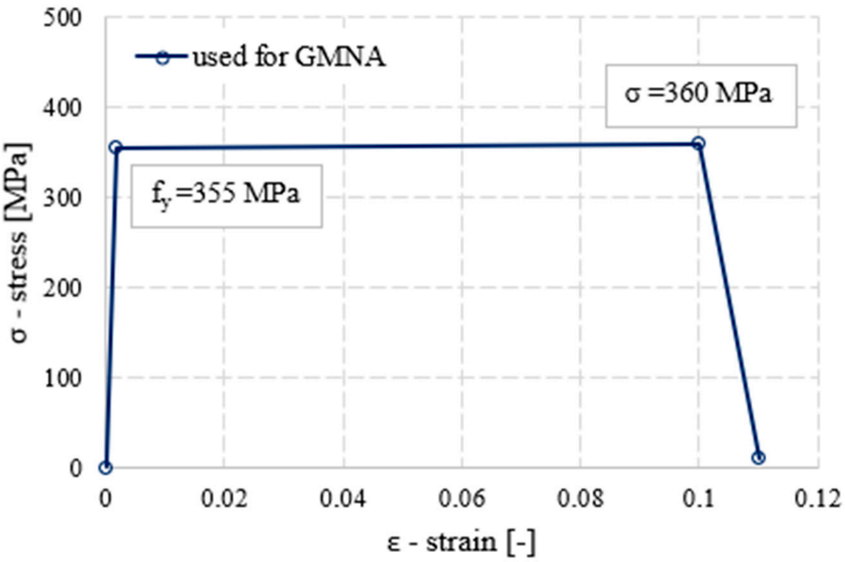

Figure 10. Multilinear stress–strain curve used in the non-linear analysis.

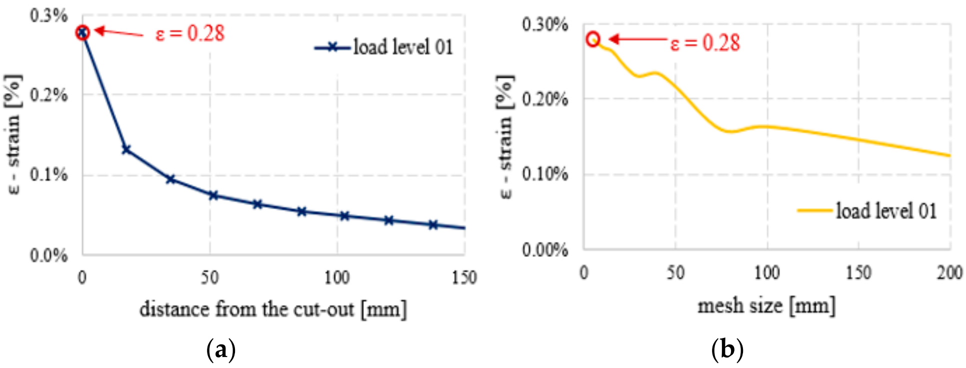

Figure 11. Strains at design load level, (**a**) depending on the distance from the cut-out, (**b**) maximum values depending on the mesh size.

4.3. Non-Linear Analysis—Evaluation of Plastic Strains

To determine how large is the real utilization ratio of the structural detail, the ultimate load is also determined by geometrical and material non-linear analysis. In the linear analysis, internal forces of M = 360 kNm and V = 72 kN were applied. Both values are uniformly increased within the non-linear analysis until the ultimate load of the structural detail is reached. The result of the GMNA analysis is the load–deflection path, as shown in Figure 12, which characterizes its structural behaviour and defines the ultimate load capacity. The results are evaluated at five different load levels (load levels 01–05) to demonstrate the failure mode and the development of the plastic zone. The analysed load levels are plotted on the load–displacement diagram in Figure 12, which are taken in quasi-equal load increments. Within the evaluation process only load levels 01 and 05 are explicitly used, the others serve as demonstrations for the plastic zone development. On the vertical axis, the load multiplier is presented corresponding to the analysed bending moment and shear force combination. The horizontal axis presents the calculated displacements at the end of the cross-girder in the vertical direction.

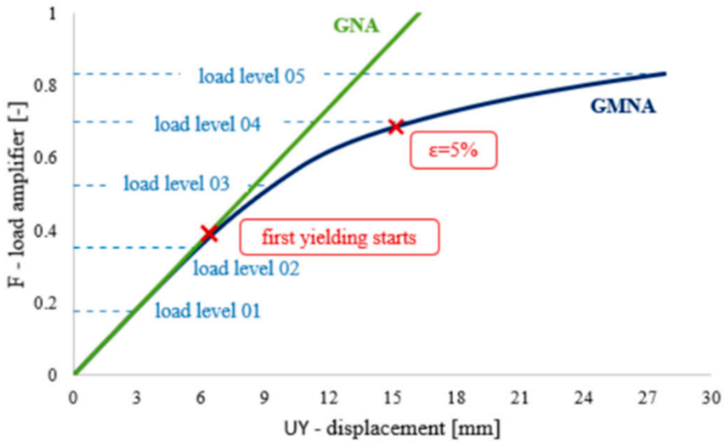

Figure 12. Load levels plotted on the load–displacement diagram.

At the highest load level (load level 05), the structure fails in the geometrical and material non-linear analysis by a load amplitude of 0.83, which value corresponds to M = 1495 kNm and V = 300 kN. The diagram also presents the load level, when the maximum strains reached 5%, which is the allowed strain limit in prEN 1993-1-14 [1] for welded plated structures. The end of the linear part of the load–displacement diagram is also determined by the comparison of the results of a GNA (geometrical non-linear analysis) and a GMNA (geometrical and material non-linear analysis). Figure 13 shows the obtained failure mode and the von Mises stress diagram, referring to a plastic failure of the cross-girder web.

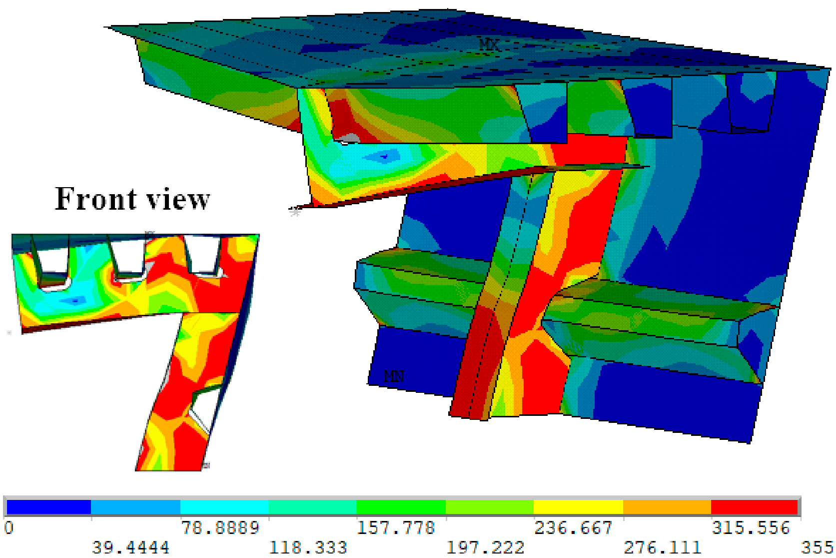

Figure 13. Failure mode (3D and front view)—von Mises stress distribution (units in (MPa)).

In the non-linear analysis, when the stress reaches the yield strength in the stress concentration zone, the material yields, the stresses will be redistributed, and the area of the yielding zone increases by the applied loads. Figure 14 shows how the stresses along the evaluation path change at different load levels. The diagram contains the results of the linear and non-linear analysis as well. The plots are scaled to 355 MPa, as the yield strength of the applied steel material. The diagram shows that for load level 05, the yield strength is reached practically along the entire evaluation path, which shows a large plastic reserve of the structural detail. Based on the non-linear analysis the resulting load capacity is equal to load level 05. It means the utilisation at load level 01 is (load level 01)/(load level 05) = 0.24 (24%). This value refers to 0.24 × 355 MPa = 86 MPa to be considered in the utilization ratio calculation if the linear analysis would be used. This value is significantly lower than the obtained stresses in load level 01 presented in Figure 14.

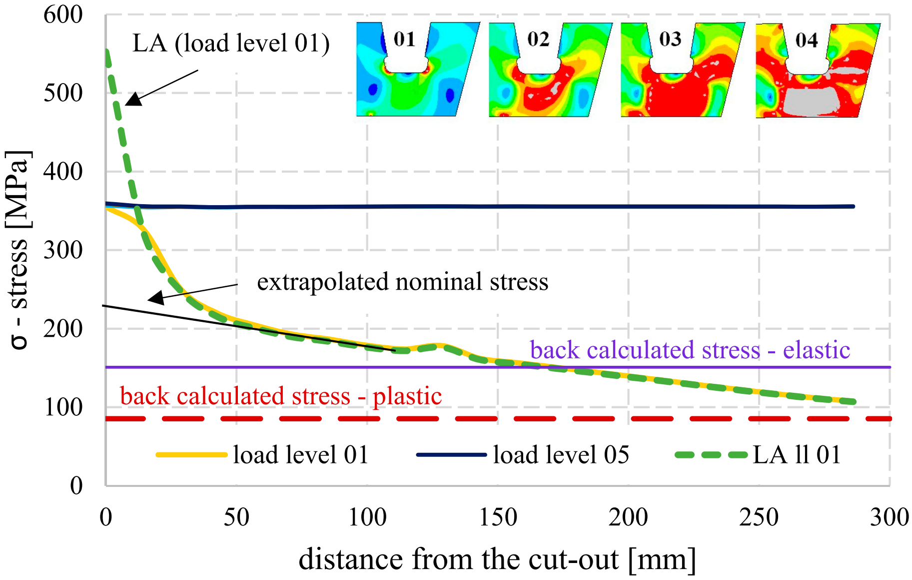

Figure 14. von Mises stresses obtained along the evaluation path (mesh size 15 mm).

The reason for it is the significant plastic reserve of the analyzed structural detail. If the ultimate load was considered as the end of the linear part of the load–displacement diagram presented in Figure 12, the first yielding starts at the load level of 0.39, which corresponds to a utilization ratio of 51% for load case 01. This utilization ratio would refer to a nominal stress of $0.51 \times 355 = 182$ MPa, which would be smaller than the extrapolated nominal stresses based on the linear analysis presented in Figure 12.

This result means the extrapolation of the nominal stresses based on the linear analysis and neglecting the stress concentration would lead to safe side resistance. However, to ensure a safe side design the maximum plastic strains should be also evaluated, to prove the accuracy of this design method. Figure 15 shows the strain distributions along the evaluation path for the five load levels based on the distance measured from the cut-out point.

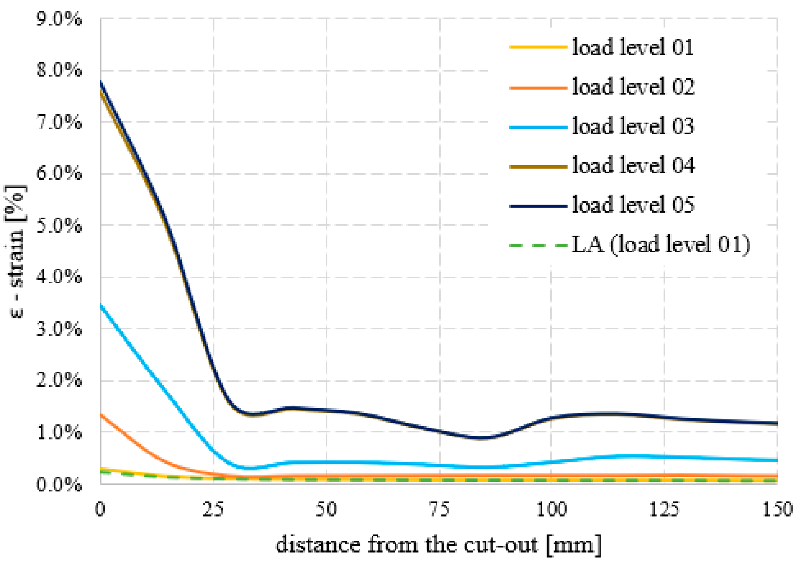

Figure 15. Strains obtained along the evaluation path for the four load levels tested.

It can be seen, that for load level 01 all the obtained strains are smaller than 0.28%; even for load level 05, the maximum strains do not reach the value related to the damage of the steel material. As the value of the maximum computed plastic strains is significantly dependent on the applied mesh size, Figure 16 shows the maximum obtained plastic strains within the stress concentration zone (at the end of the evaluation path). The results show the obtained maximum values are significantly smaller than the maximum allowed plastic strain the material can perform, so there is no crack propagation risk due to the plastic strains.

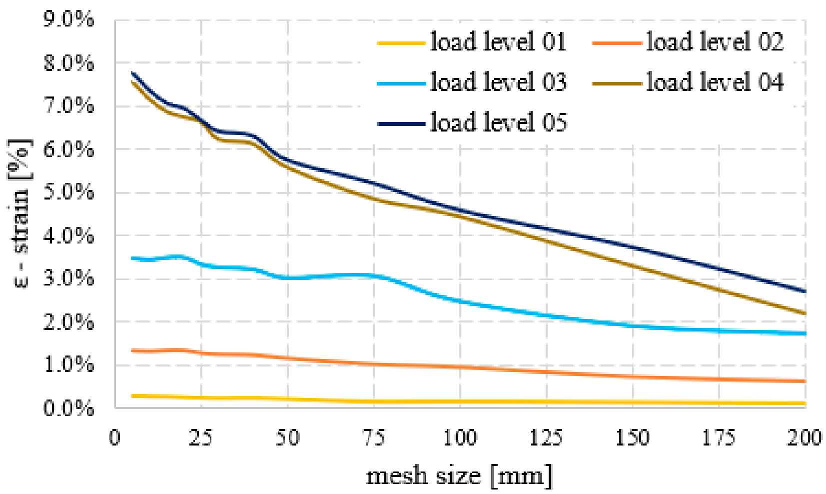

Figure 16. Maximum strains in the stress concentration zone depending on mesh size.

5. Summary and Conclusions

In the present paper, a numerical study is introduced to investigate the stress concentration zones and their consideration in the design check using linear elastic analysis. The proposed evaluation methodology of the stress concentrations is presented based on the European pre-standard prEN 1993-1-14. The current research aimed to demonstrate the application of the Eurocode-based general design rules, and it gave as an extension of the code rules a general evaluation strategy for the stress concentration zones, which can be followed in the design practice.

In this paper, first, the proposed evaluation methodology is introduced, and its application is presented on one structural detail having significant stress concentration. On the developed numerical model linear and non-linear analyses are performed and based on the computed stresses and strains the proposed evaluation methodology is introduced. The same methodology can be used for other structural details similarly.

Based on the numerical study the following conclusions are drawn:

- In the stress concentration zone at first the mesh-independent stresses should be determined by a mesh sensitivity study, eliminating the numerical singularities from the obtained results.
- For the analysed structural detail representing typical structural solution for bridges, the extrapolated nominal stresses should be determined, which can be used based on linear analysis to perform the design check.
- The maximum value of the mesh-independent stresses should be also determined and evaluated. If the maximum value exceeds the yield strength, the maximum plastic strains should be determined and evaluated.
- The investigations prove the maximum calculated strains in the stress concentration zones are significantly smaller than the capacity of the steel material for the analysed case, therefore, there is no risk of fracture because of the stress concentration.
- However, the maximum strains are significantly dependent on the applied mesh size. If rounded corners are applied in the numerical model, the maximum strains are tending to a maximum value, and do not go to the infinite by decreasing the mesh size.

- Based on the non-linear analysis and by determining the load–deformation path of the analysed structure, the application of the extrapolated nominal stresses gives safe side results by the utilization ratio calculation.

Similar studies are going on with different structural details to make more general conclusions regarding the applicability of the presented design method and the negligibility of the non-linear analysis.

Author Contributions: Conceptualization, L.D.; Methodology, B.K. and L.D.; Software, E.B.; Investigation, E.B. and B.K.; Writing—original draft, E.B., B.K. and L.D; Writing—review & editing, E.B. and B.K. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: There are no data available to this manuscript.

Acknowledgments: The presented research program has been financially supported by the Grant MTA-BME Lendület LP2021-06/2021 "Theory of new generation steel bridges" program of the Hungarian Academy of Sciences, which is gratefully acknowledged.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. *prEN 1993-1-14*; Eurocode 3: Design of Steel Structures—Part 1-14: Design Assisted by Finite Element Analysis. European Union: Brussels, Belgium, 2024; under publication.
2. Oldal, I. *The Practice of Applying the Finite Element Method in Mechanical Engineering Design*; Hungarian Chamber of Engineering, Division of Mechanical Engineering: Budapest, Hungary, 2015. (In Hungarian)
3. Nemade, A.; Shikalgar, A. The mesh quality significance in finite element analysis. *J. Mech. Civ. Eng.* **2020**, *17*, 44–48. [CrossRef]
4. Li, Y.; Wierzbicki, T. Mesh-size effect study of ductile fracture by non-local approach. In Proceedings of the SEM Annual Conference, Albuquerque, NM, USA, 1–4 June 2009.
5. Liu, W.; Geni, M.; Yu, L. Effect of mesh size of finite element analysis in model analysis for periodic symmetric struts support. *Key Eng. Mater.* **2011**, *462–463*, 1008–1012. [CrossRef]
6. Raut, P. Impact of mesh quality parameters on elements such as beam, shell and 3D solid in structural analysis. *Int. J. Eng. Res. Appl.* **2012**, *2*, 99–103.
7. Liu, Y. Choose the best element size to yield accurate FEA results while reduce FE models' complexity. *Br. J. Eng. Technol.* **2013**, *1*, 13–28.
8. More, S.T.; Bindu, R.S. Effect of mesh size on finite element analysis of plate structure. *Int. J. Eng. Sci. Innov. Technol.* **2015**, *4*, 181–185.
9. Wang, E. Thin-walled structure simulation. In Proceedings of the International ANSYS Conference, Pittsburgh, PA, USA, 2–4 May 2006.
10. Wang, Y.B.; Lyu, Y.F.; Li, G.Q.; Liew, J.Y.R. Behavior of single bolt bearing on high strength steel plate. *J. Constr. Steel Res.* **2017**, *137*, 19–30. [CrossRef]
11. Vértes, K. Investigation of Steel Joints with Non-Overhanging Connection Plates, 3D Joints and Chain Elements. Ph.D. Thesis, Department of Structural Engineering, Budapest University of Technology and Economics, Budapest, Hungary, 2006.
12. Langer, K. Comparative Analysis of Different Analysis Types Performed on Shell Models. Master's Thesis, Department of Structural Engineering, Budapest University of Technology and Economics, Budapest, Hungary, 2018.
13. *CEN/TR 1993-1-141*; Eurocode 3: Design of Steel Structures—Part 1-141: Background and Explanations on EN 1993-1-14 Design Assisted by Finite Element Analysis. European Union: Brussels, Belgium, under publication.
14. Ansys Inc. *ANSYS®v19.0 [Software]*; Ansys Inc.: Canonsburg, PA, USA, 2021.