Danish University Colleges

How to deal with the annoying Hot Spots in FEA

Svenninggaard, Jon

Publication date:
2017

Link to publication

Citation for published version (APA):
Svenninggaard, J. (2017, Mar 3). How to deal with the annoying Hot Spots in FEA.

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

• Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
• You may not further distribute the material or use it for any profit-making activity or commercial gain.
• You may freely distribute the URL identifying the publication in the public portal.

Download policy
If you believe that this document breaches copyright please contact us providing details, and we will remove access to the work immediately and investigate your claim.

Download date: 26. jun., 2019
How to deal with the annoying Hot Spots in FEA

Jon Svenninggaard
Department of Mechanical Engineering
VIA University College, Denmark

February 20, 2017

1 Introduction

Often, when I talk to fellow engineers one of the most common question I hear is; How do we deal with the annoying “Hot Spots” in our solid structure Finite Element Analysis (FEA) and how do we justify the stresses in those areas?

A "Hot Spot" is in this context a geometric singularity where high stresses are confined to a small area. This area stands out as it mostly represents stresses that exceed the allowable. The singularity may come from boundary conditions where the transition between stiffness is high, it may come from contact problems, it may arise from sharp edges where the high stressed area is confined to the elements near the corner, or it can emerge from discretization error of the mesh. The mesh generation typically results in that most fillets in the 3D CAD (Computer Aided Design) model, is simplified as perfectly sharp corners. We all know that perfectly sharp corners and infinite stiff structures do not exist in real life. The first two problems are exemplified in Fig. 1. Here, the first image shows the boundary conditions. The bracket is fixed on the upper surface and a load of 1000 N is applied at the other end. Initially, using an element size of 10 mm and 8-noded solid elements with 24 degree of freedom (DOF), the stresses are shown in the middle image. After a mesh refinement, the stresses in the inside corner increase and would increase even further by additional mesh refinements. Furthermore, it can be seen that there is an increased stress near the edges where the fixed support is located. So, can we just disregard these areas where the stresses are high or do we need to change the geometry, boundary conditions or use a completely different approach?

Initially, we need to understand that most Finite Element Analysis is performed as linear elastic. This means that the displacements, stresses and strains are doubled if the load is doubled. I.e. there is a clear linear correlation between those field quantities. This does not relate to the real world where the stress -
strain relationship in most ductile metallic materials is only linearly related until a certain point. After this, yielding and plasticity are the governing parameters and the linear force - displacement relationship is no longer valid. Thus, when we see stresses that are far beyond the ultimate limit strength, often this is not the entire truth and a singularity or "Hot Spot" may be present. Moreover, sharp corners in real life are always rounded. Even if it cannot be seen by the naked eye it will seem rounded if observed through a microscope. This even applies for most sharp cracks. However, even if smaller loads are applied to very sharp corners or cracks, the geometry alters due to crack tip localized plasticity and the stresses distribute as if the corner was rounded. Nonetheless, it is highly important to estimate the strain or stress condition at these locations correctly if fatigue loading is involved. If fatigue loading is involved, the plastic strain due to the presence of sharp corners can easily lead to the formation and propagation of cracks, which ultimately could lead to a complete loss of bearing capacity in the structure. This also applies if static loadings are involved as the stress concentration factor could lead to stresses above the ultimate limit stress of the material and a sudden rupture might occur.
2 How to deal with it?

The first thing to consider when a "Hot Spot" is observed is obviously to carry out a mesh convergence study. This would give the engineer information about the stress gradient in the area, and if appropriate elements and element sizes have been used! One method is to utilize one of several adaptive meshing techniques; These are the \( p \)-refinement technique that changes the polynomial order of the element. The \( r \)-refinement technique that relocates elements and the mesh. Or the \( h \)-refinement technique that refines the mesh in areas of high stress gradients. Even combinations hereof exists as for example the \( hp \)-refinement technique. Some of these techniques are usually build into commercial finite element software packages. The \( h \)-refinement and \( p \)-refinement techniques utilizes the posteriori Zienkiewicz - Zhu (ZZ) error estimate \( [1] \) that is determined as:

\[
||U||^2 = \sum_{i=1}^{m} \int \{\epsilon\}_i^T [E] \{\epsilon\}_i dV
\]  

(1)

Here, \( m \) is the number of elements in the structure or area of interest, \( [E] \) is the constitutive matrix and \( \{\epsilon\} \) is the strain vector. The global energy error norm in the structure is determined as:

\[
||e||^2 = \sum_{i=1}^{m} \int (\{\epsilon^*\}_i - \{\epsilon\}_i)^T [E] (\{\epsilon^*\}_i - \{\epsilon\}_i) dV
\]  

(2)

Here, \( \{\epsilon^*\} \) represents the smoothed strain field over the entire structure. Thus, the global energy error norm gives a measure of the variation of energy within the structure. This is then utilized to find the relative error in the structure as:

\[
\eta = \left[ \frac{||e||^2}{||U||^2 + ||e||^2} \right]^{1/2}
\]  

(3)

This relative error \( \eta \), can take values in the range; \( 0 > \eta > 1 \). Consequently, a good discretization is usually obtained with \( \eta \leq 0.05 \). For the case of the \( h \)-refinement technique this is illustrated in Fig. 2. Here, the plate is loaded by an evenly distributed load on the top edge and fixed along the edge A-B. Using adaptive meshing with the \( h \)-refinement technique, the mesh is refined around the corners. With increasing mesh density, the stress at these points will tend towards infinity. Already after the first mesh revision, the engineer would realize that he or she instead, should consider changing the boundary conditions, or simply disregard these areas from the stress analysis.

The example illustrated in Fig. 2, is further visualized in Fig. 3. Using the commercial software Ansys Workbench, the difference between the left boundary being completely fixed and it being subjected to a different boundary condition is easily seen. The analysis shown in Fig. 3 a) to c) never converges! The mesh

...
density is increased in the corners by each iteration and the stress increases. In Fig. 3 d) and e). The analysis converges within a few iteration. This is illustrated in f). The relative error $\eta$ in the model cannot directly be plotted, but it can be scripted quite easily in most FE packages.

If we turn our attention towards other methods and approaches, a lot of work has been committed to finding methods to deal with weld details and relating them to a certain Fatigue class (FAT) using the FEM. Most used is probably the "Nominal stress method" which relates a number of structural details to certain
fatigue classes or SN - curves. The method has among others been adopted by Eurocode 3, part 1-9, [2] and a series of other 3rd party organizations. This method however, is not directly usable for the above discussed singularities as no listed structural details exists to quantify the various singularities that may emerge in the FEA.

A more specialized method that has been standardized by the International Institute of Welding [3] and DNV [4] is the "Structural Hot Spot method". This method can be used either with physically mounted strain gauges or the Finite element method. The method is based on that strains or stresses are evaluated at certain distances from the weld toe and extrapolated here to. Then, the stress or strain level of the detail is compared to a certain FAT class. This method can prove quite handy for shell models. For shell models, the stress at sharp corners or transitions between profiles or plates, can be evaluated by extrapolating stresses into the corner. Thereby the "Hot Spot" can be quantified and the stresses can be shown to have an absolute value. Nonetheless, the calculated stress should be considered carefully as other factors might influence.

The newest and in my opinion, most promising method with the finite element method, is the "Effective Notch method” [3]. Here, the corner or weld detail in the 3D CAD model is modeled using a fillet of 1 mm and in the finite element software a max element size of 0.25 mm in this area is applied. Then the maximum principal stress is compared to a FAT 225 curve [5] [6]. Thus, for static applications the computed max stress is compared to the ultimate limit strength of the material. The drawback is that it may be computationally too expensive to use this element size in the entire model. Therefore, it should be considered to use a sub-model of the area of interest.

As mentioned earlier, "Hot Spots" are also common in contact problems. One example is illustrated in Fig. 4. Here, the bolted connection, comprised of frictionless contacts between multiple parts, shows very high stresses in a localized area. Thus, it can be considered a "Hot Spot". The high stresses typically originate from non-conforming meshes between the parts, or from a single node connected through contact elements with multiple nodes on the counterpart. A method to justify the stresses in this area could be to use a finer mesh on the parts in contact. This however, might be impossible due to computational expense. Moreover, the area could be used in a sub-model, where the displacements are transferred from the global model. In this sub-model, the mesh can easily be refined and or an elastic - plastic material model can be used. Sometimes, this is not possible. In this case it might be reasonable to extract the contact forces and analyze the connection with traditional analytical methods. These methods could follow standards as for example Eurocode 3, part 1-8 [7] or VDI 2230 [8] or other.
3 Conclusion and discussion

This small guide introduces a few methods to handle singularities often referred to as "Hot Spots" in finite element analysis. The first step should always be to conduct a mesh convergence study in order to make sure that the correct element sizes and types have been chosen. Automatic adaptive meshing techniques can be used. However, these methods should be handled with care. Moreover, several techniques are described that have been developed for fatigue evaluation of welds. Some of these methods can be utilized to estimate the stress state near corners or areas where the stress gradient is high. Often, contact problems exhibit areas where localized stresses are very high. Here, a sub-model or local mesh refinement might alleviate the problem. Otherwise, the contact forces should be extracted and the contact should be analyzed using analytical methods.
References


