Revisiting Stress Singularity / Concentration Examples with COMSOL Multiphysics®

Ivar KJELBERG
Centre Suisse d'Électronique et de Microtechnique,
CSEM SA, Neuchâtel, Switzerland

INTRODUCTION: While following Tony ABBEY's interesting NAFEMS talks[1], I started wondered why my COMSOL models look different? Here I focus on Stress Singularity (SS) and concentrations (SC) that come in two basic flowers: SS of numeric origin (mostly non-physical) and "true" SC, both are essential to acknowledge.

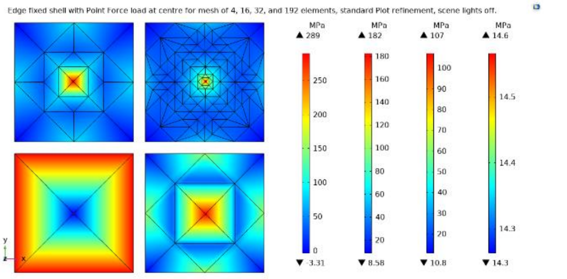

Figure 1. Central Point load SS on a Shell with increasing mesh elements.

MODEL 1: NUMERIC STRESS SINGULARITY (SS)

The App model above is built with standard COMSOL "Solid" physics settings of a 3D Shell with few mesh elements. The Point load shows up clearly as a ***numeric stress singularities***, and even if we increase the number of mesh elements, the stress singularity becomes only of smaller area but with higher stress values, as expected. *Why:* because FEM is based on physics applied to Domains, with associated "Boundary Conditions" applied to the Boundaries, **not** to lower-dimensional entities. The Force load used in Solid is in fact a *Force Density [N/m^2] distributed over a Boundary*. Any force applied to an Edge or a Point has no associated area, apart from the direct adjacent-mesh elements, hence if these become smaller the Force density applied will only increase, endlessly (also check the St-Venant principle). Numerical SS "hot-spots" appear easily for welded beams and structures with sharp corners, adding fillets will often improve your results, but adds RAM demand.

Still my results look "smoother" than those obtained out of the box from older FEM programs and ref. (1). My explanations: COMSOL uses by default ***Quadratic Discretisation*** and Plot Quality Smoothing: which corresponds to higher order mesh elements and added plot rendering, the latter might be misleading, though. ***Hint****: Test the Quality settings and try to turn off "Scene Lights" to improve the colour luminosity of your plots*.

MODELS 2-4: STRESS CONCENTRATION (SC)

SC show up when we have shape changes for loaded parts, as for a thin slender beam attached to a bulk part, even if we add an elliptical fillet to smoothen the transition a question arises which radii to use ? The older FEM specialists would dig into their references such as (2), but one may also use COMSOL to analyse and optimise the shapes of the junction region:

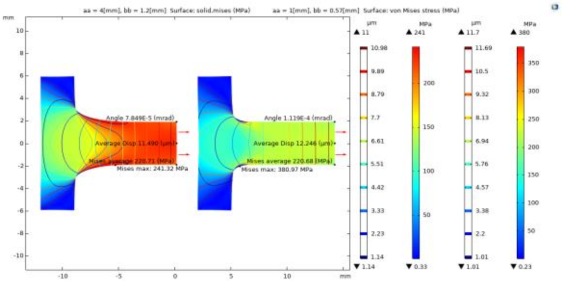

Figure 2. Beam SC in Traction Fx, elliptic transition.

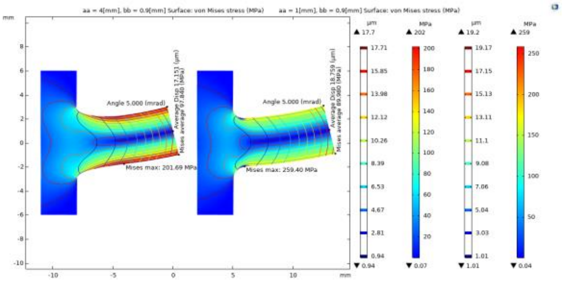

Figure 3. Beam SC in Moment Mz bending, elliptic transition.

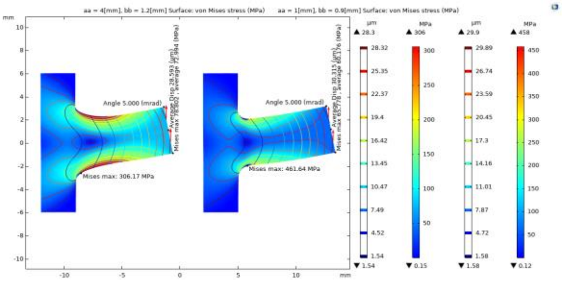

Figure 4. Beam SC in Force bending Fy, elliptic transition.

The three 2D solid "Plane strain" App models above are fixed to the left, and the right side undergoes traction ***Fx***, Moment bending ***Mz*** and respectively force bending ***Fy*** with quite different SC's, *also note the very different stress distributions!* The two elliptic fillets of radius $\boldsymbol{aa_x}$ and $\boldsymbol{bb_y}$ are scanned by a "parametric sweep" for the former and Nelder-Mead "optimised" for the latter. ***Hint:*** *To get a fine mesh at the high stress regions I used the COMSOL solver Adaptation Mesh refinement in 3 steps by using the strain energy variable "**solid.Ws**" as "Error indicator", and 0.2-0.4 as "Fraction of elements" limit, and **not** the default **L2** norm.*

CONCLUSIONS:

If one can only try to avoid numerical SS by changing the Boundary load types, the true SC might efficiently be minimised through an optimisation with COMSOL. The results from the loaded flexible beam is according to the standard references (2) : an elliptical transition with a beam thickness to ellipse radius $\boldsymbol{aa_x}$ ratio of ≥ 1 gives less than 10% stress increase for the ***Fx*** & ***Mz*** load cases.

REFERENCES: see https://www.comsol.ch/community/exchange/821/

1. Tony ABBEY, see www.nafems.org and www.fetraining.net/ .
2. W.D. PILKEY, "Peterson's Stress Concentration Factors", J.Wiley & Sons, 1997.
3. Mph model files: *RSCE_Ikj_PL, RSCE_Ikj_Fx, RSCE_Ikj_Mz, RSCE_Ikj_Fy* .

Excerpt from the Proceedings of the 2020 COMSOL Conference