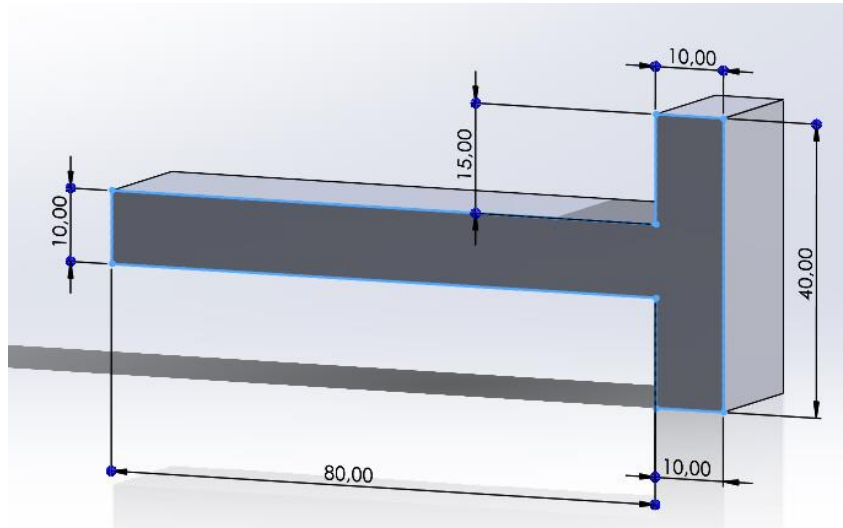
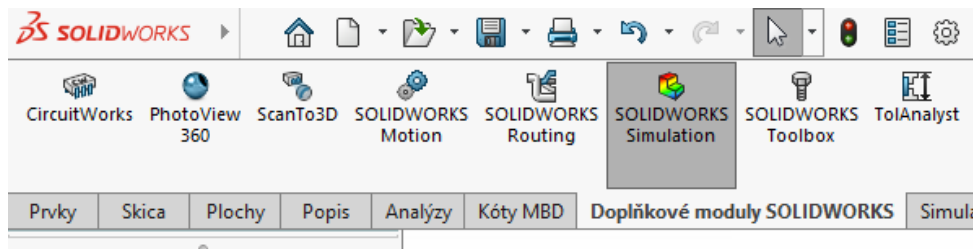


Tension simulation

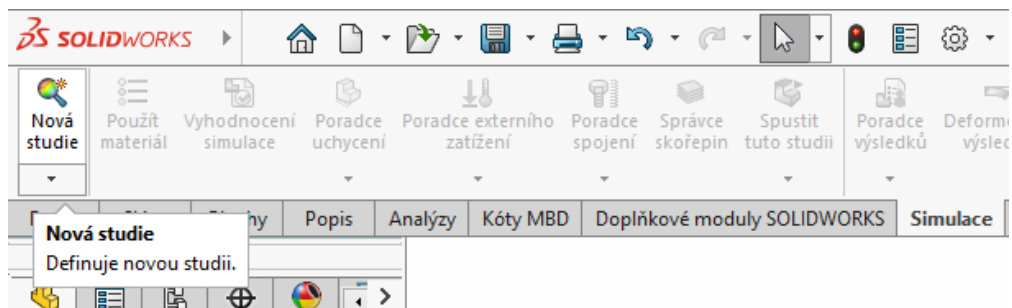
1. Create a simple part on which we will apply the tension analysis. Follow this figure and extrude by 20 mm.



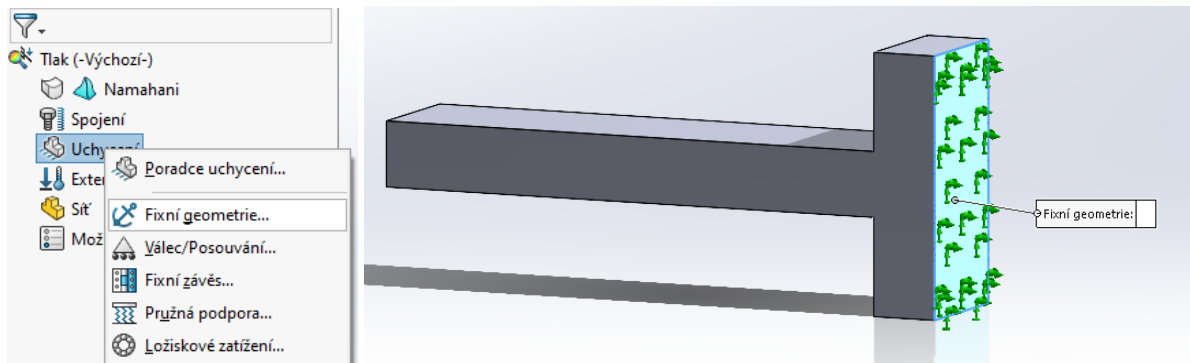
2. Activate the SOLIDWORKS simulation add-in.



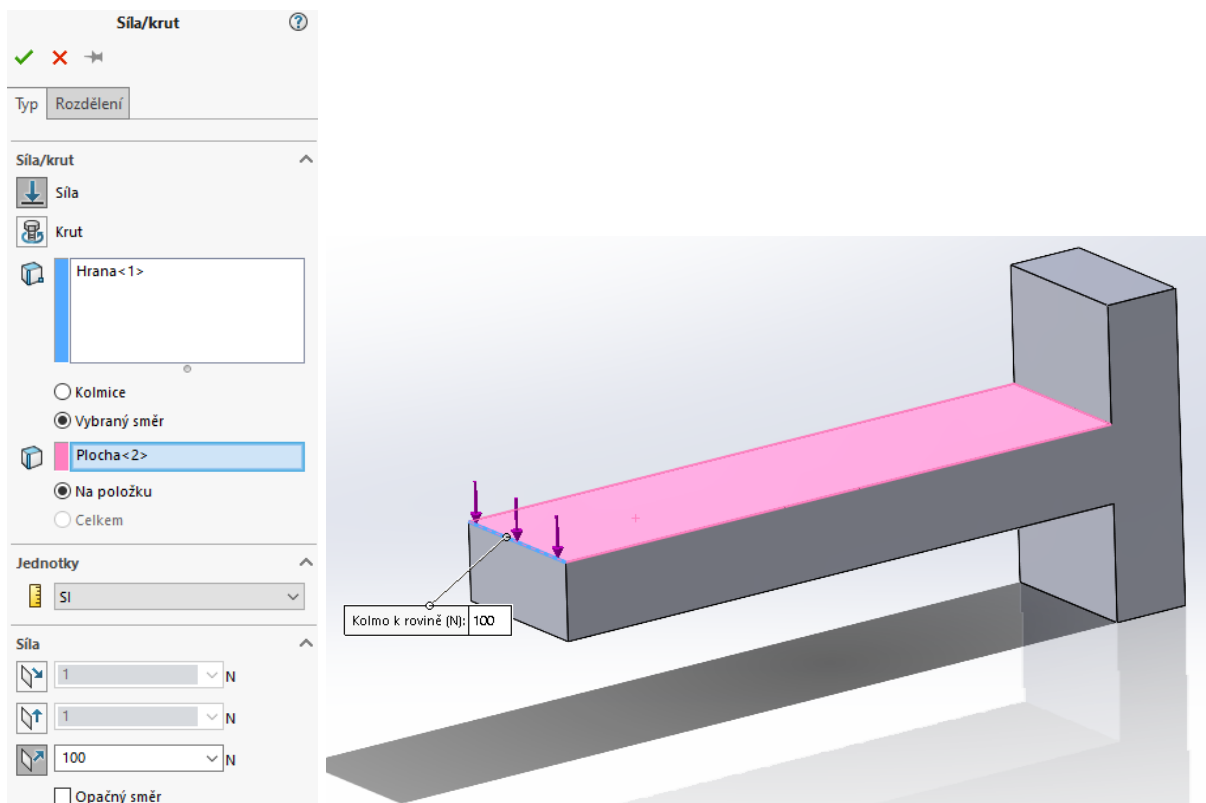
3. In the simulation tab start a new study.



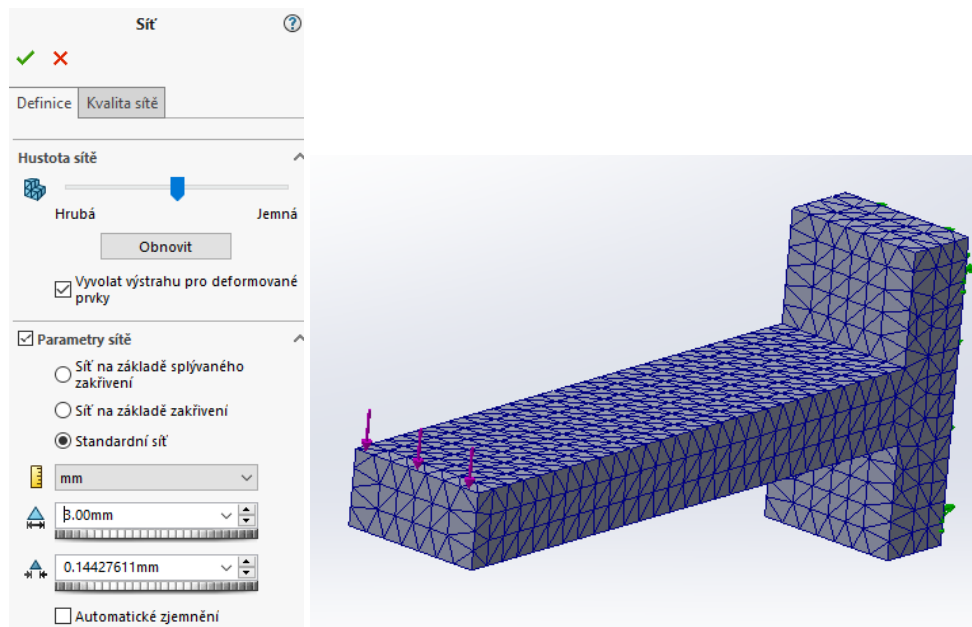
4. Add a fixed geometry – a reference to a plane which is firmly attached.



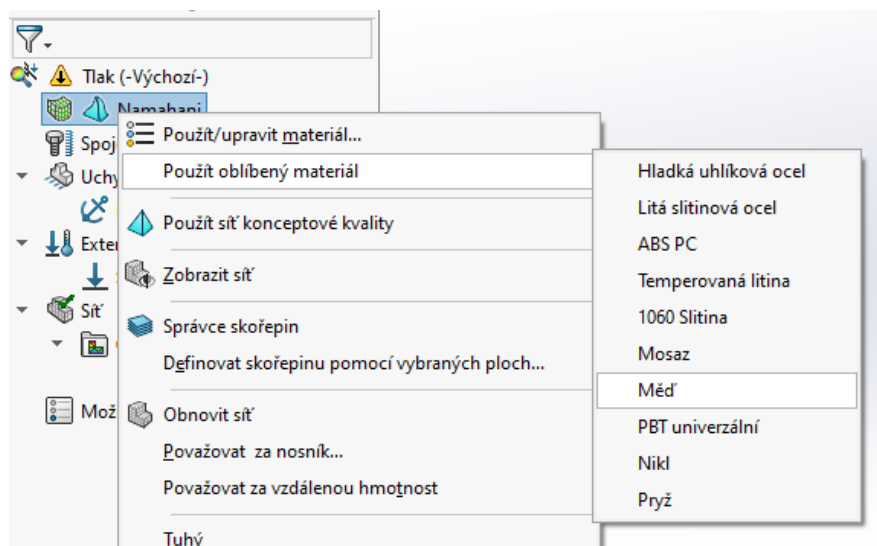
5. Add external stress of the force type to the end edge of the part and also add the direction which will be the top plane of the part. Set the amount of force to 100 N.



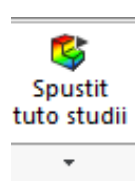
6. Add creation of polygon network to count the tension. Set the global size of polygon to 3 mm and run the calculation.



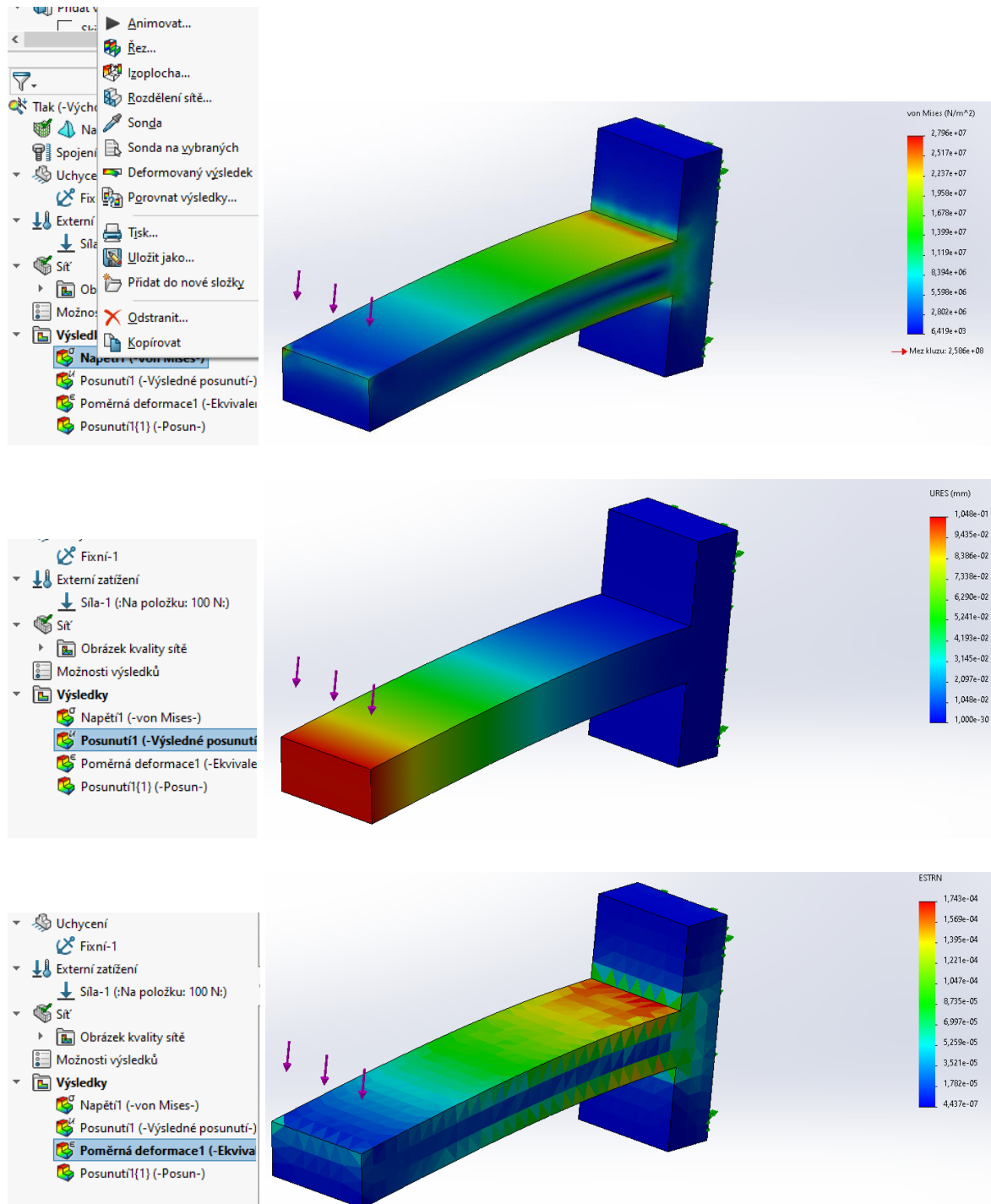
7. Add material from tension menu. Select copper.



8. Run the calculation of the study.



9. In the tree of the study you can switch between different types of computed tension metrics.



10. Try changing various parameters of the study – force applied, polygon network density and so on. Observe the differences that these changes make.