KISSsoft
Calculation programs for machine design

KISSsoft, FEM based root stress calculation

Application example jack up gears, plastic gears and metallic gears with grinding notches

EES KISSsoft GmbH
Hauptstrasse 7
6313 Menzingen
Switzerland

Tel: +41 41 755 33 20
h.dinner@EES-KISSsoft.ch
www.EES-KISSsoft.ch

SHARING KNOWLEDGE

1 Document information

1.1 Document change record

Revision	Date	Author	Comments
0	14.2.2014	HD	Original document

1.2 Table of content

1.3 References

[1] KISSsoft release 03-2017A
[2] A. N. Montestruc, Allowable Contact Stresses in Jacking Gear Units Used in the Offshore Industry, Gear Technology, May 2010

2 KISSsoft release 03-2017

2.1 Tooth root stress calculation

In KISSsoft release 03-2017, a tooth root stress calculation using a 2D FEM model of the virtual spur gear is implemented.

The gear root strength is typically calculated along ISO 6336. In this approach, a virtual spur gear is used in case of helical gears. This means that the properties of a helical gear are first converted into a corresponding spur gear and then the strength rating is done for this spur gear. Some properties of the calculation are:

- Tooth root stress is calculated in one specific section of the root, where tooth thickness and radius of curvature is calculated based on a theoretical hob (ISO 6336) or a gear shape cutter (AGMA 2101).
- The calculations gives relatively good results for gears with standard tooth profile.
- The results are not accurate in case of:
  - Gears with pressure angle $a_n < 17.5^o$ or $a_n > 22.5^o$
  - Gears with high helix angle $b > 25^o$
  - Gears with big profile shift coefficient
  - Gears with tooth modifications
  - Gears with high contact ratio (deep tooth profiles)
  - Gears with undercut or grinding notch
  - Internal gears by AGMA (no formulas for the geometry factor)
  - Non-involute gears (such as cycloids)

To check the tooth root stress by an independent approach, a FEM calculation is implemented in KISSsoft. There, the same approach as per ISO 6336 is used

- Calculation is done on the virtual spur gear
- Calculation is done with the same load and load direction as in the ISO calculation
- Stress level is checked at same position in the root

This approach allows for a direct comparison of the results from the FEM calculation with the results from the ISO calculation. The FEM calculation may therefore be combined with the ISO rating, giving safety factors.

2.2 Intended use

The new KISSsoft module should be used in case a more detailed investigation of the gear root is called for. This may be required for

- Gears that are machined e.g. on 5-axis CNC machines
- Injection molded or sintered gears
- Gears with grinding notches

2.3 Software usage

An additional tab is added in KISSsoft. There, the FEM calculation can be set up.

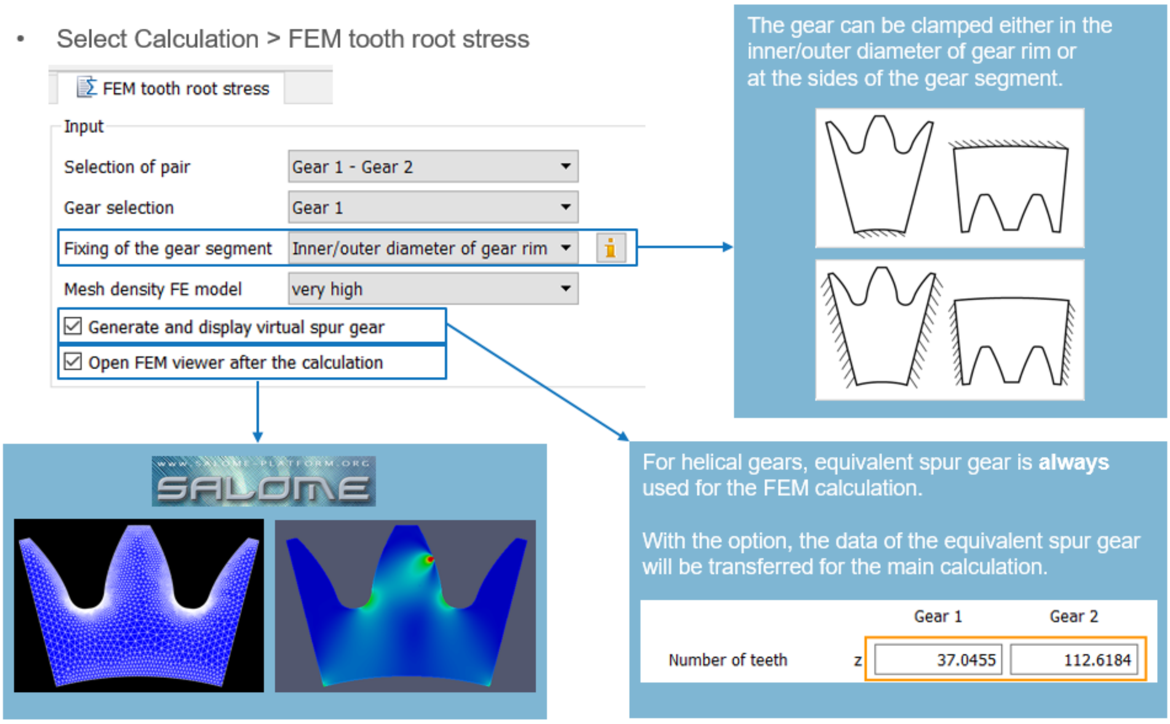


Figure 2.3-1 Set up of the calculation, boundary conditions, resulting FEM model and virtual spur gear data

3 Application example, jack up gears

3.1 Introduction

An offshore jack-up drilling rig is a barge upon which a drilling platform is placed. The barge has legs that can be lowered to the sea floor to support the rig. Then the barge can be “jacked up” out of the water, providing a stable work platform from which to drill for oil and gas. Jack-up drilling rigs were first introduced in the late 1950s. Rack-and- pinion-type jack-up units were introduced soon after that and have dominated the industry ever since. The rack-and-pinion systems used to raise and lower the rig are enormous in terms of gear pitch, or module, by gear industry standards. Quarter-pitch (101.6 module) pinions are common, with both larger and smaller teeth used. The lifetime number of cycles for these units is—by gear industry standards—tiny in that rack teeth typically have 25-year lifetime cycles measured in the low hundreds. That is off the charts for AGMA (and ISO or DIN) design rules, which draw a straight line to zero cycles for contact stress cycles less than 10,000, [2].

Figure 3.1-1 Left: Jackup platform. Middle: Rack and pinion system on leg. Right: KISSsoft rack and pinion.

It is hence of interest to directly calculate e.g. root stresses and compare them to e.g. yield strength. Furthermore, the tooth root shape of the pinion may have different forms, e.g.

- Case 1: Trochoidal fillet in case a hob with circular tip and a generating process is used
- Case 2: A full, truly circular root shape, e.g. if the gear is machined on a 5-axis CNC milling center
- Case 3: An optimized root shape to minimize root stresses, e.g. an elliptical root shape

The use of the new KISSsoft root stress calculation is shown and the three cases above are compared.

A generic load case, resulting in high root stress, is used for the below calculations.

3.2 Basic gear geometry data

A basic gear design with the below data was selected for this application example:

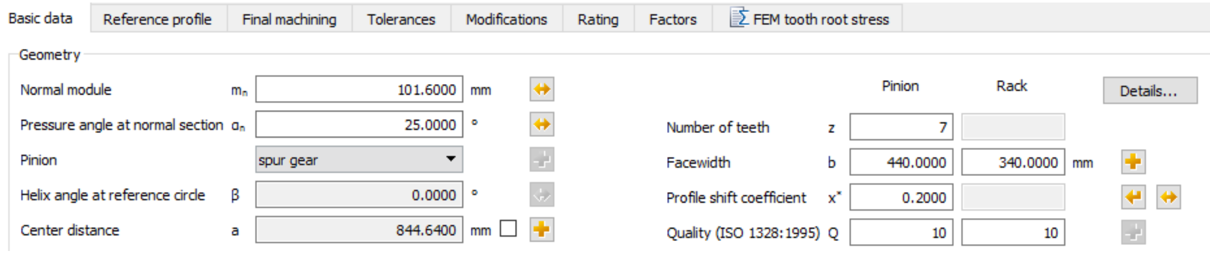


Figure 3.2-1 Basic gear parameters

From the above, the below shown gear profile results:

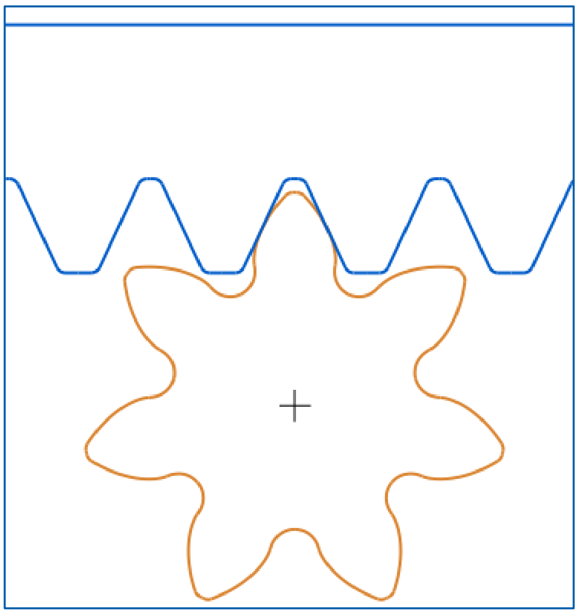


Figure 3.2-2 Left: Rack and pinion. Right: Generating cuts, pinion.

3.3 Pinion root stress for case 1 (trochoidal fillet)

Let us assume that the pinion is manufactured using a generating process. The tool has a circular tip rounding, resulting in a trochoidal fillet in the pinion root. The gear basic rack root rounding factor is $\rho fP^*=0.500$.

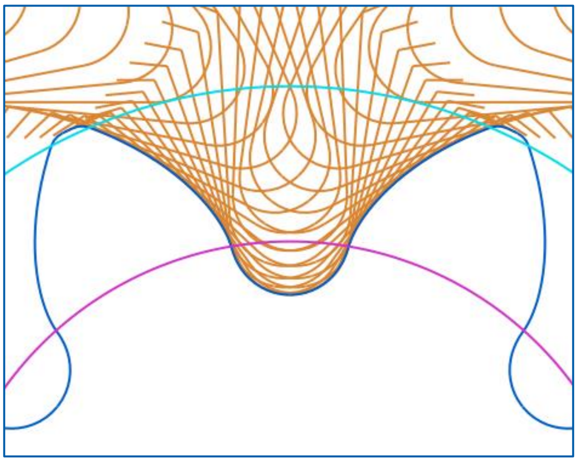


Figure 3.3-1 Root shape, generated from hob, for case 1.

We may now calculate the pinion root stress using the FEM calculation and find a tooth root stress of 693.49MPa:

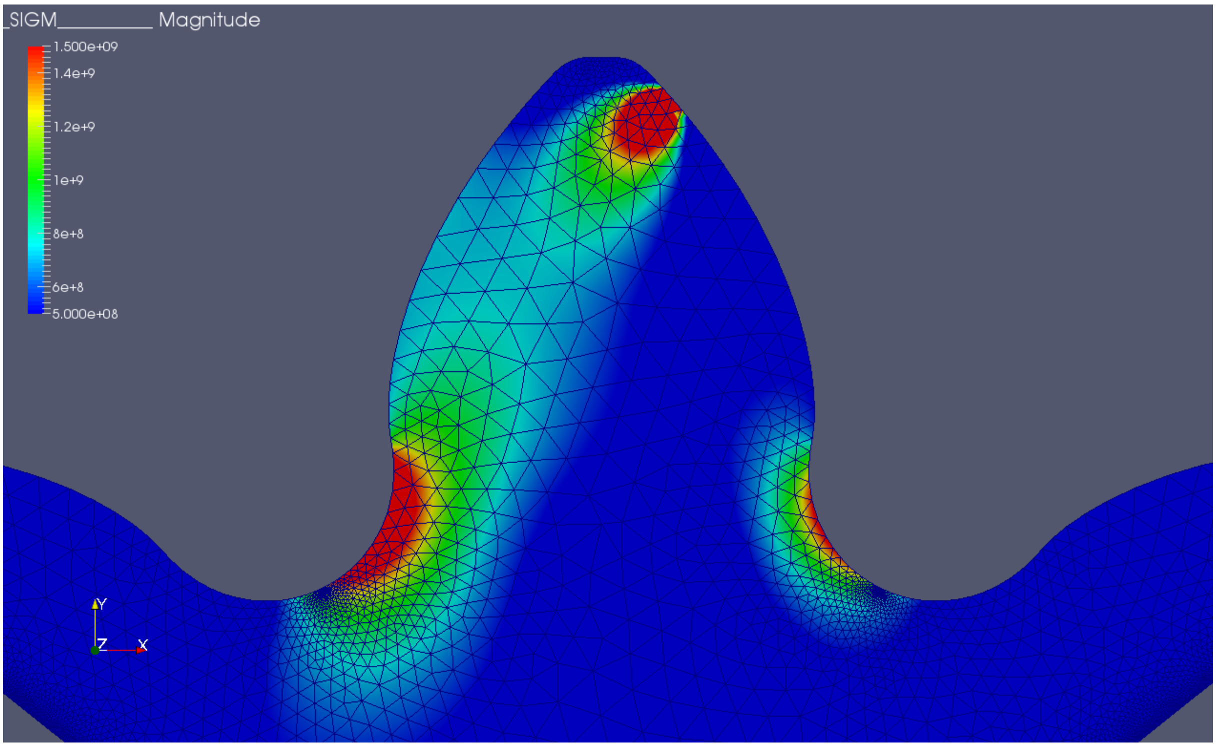


Figure 3.3-2 Tooth stress for case 1. Tensile stress on right flank, compressive stress on left flank. Scale: 500-1500MPa.

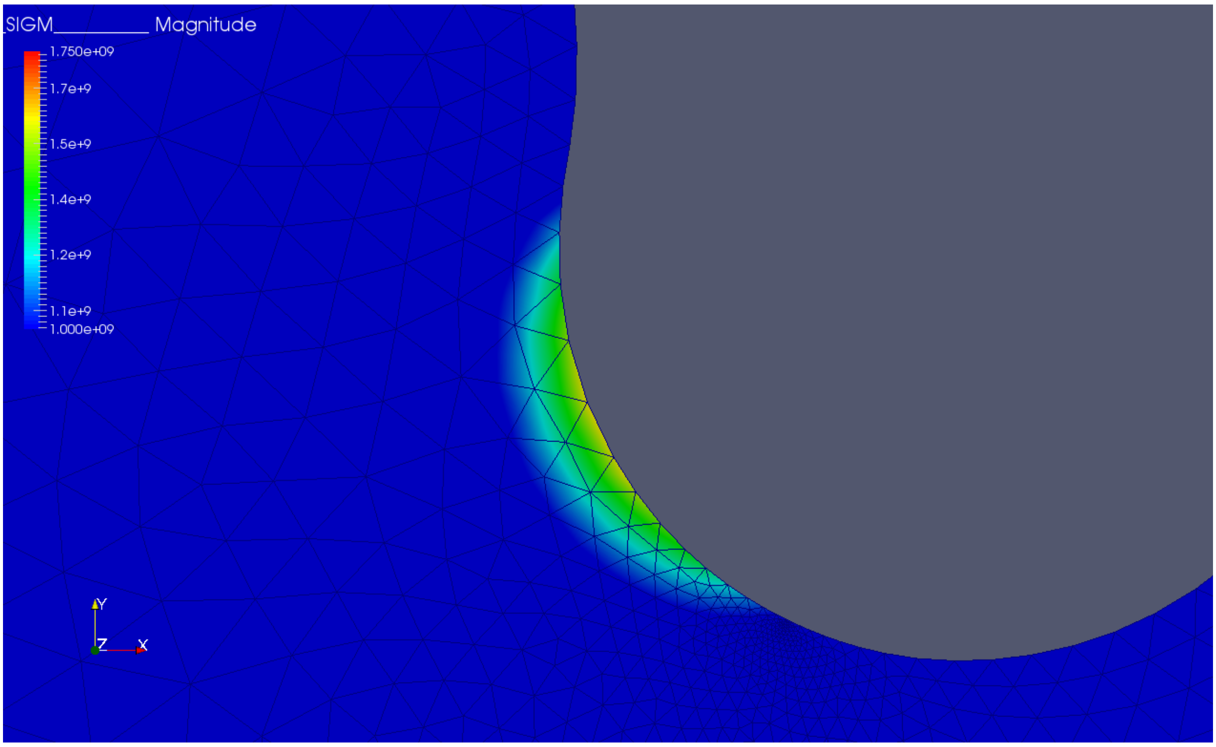


Figure 3.3-3 Tooth stress for case 1. Detail view of root form. Scale is 1000MPa-1750MPa

3.4 Pinion root stress for case 2 (circular root shape)

Let us assume that the pinion is manufactured on a 5-axis CNC milling machine and the root shape is truly circular with a radius of 0.500*101.6mm. We then find a small change in the root as shown below. There, the blue curve is the tooth root shape if the gear is manufactured by a generating process while the orange color shape is the tooth root shape for a truly circular tooth root. Note that the basic rack properties are the same in both cases.

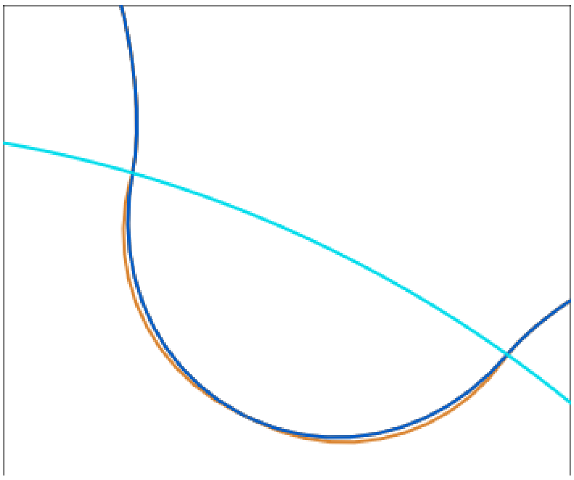


Figure 3.4-1 Circular root shape (orange) vs. trochoidal fillet (blue). Circular root shape is used in this case 2 calculation.

We then find the below tooth root stress. Note that it has increased to 728.07MPa.

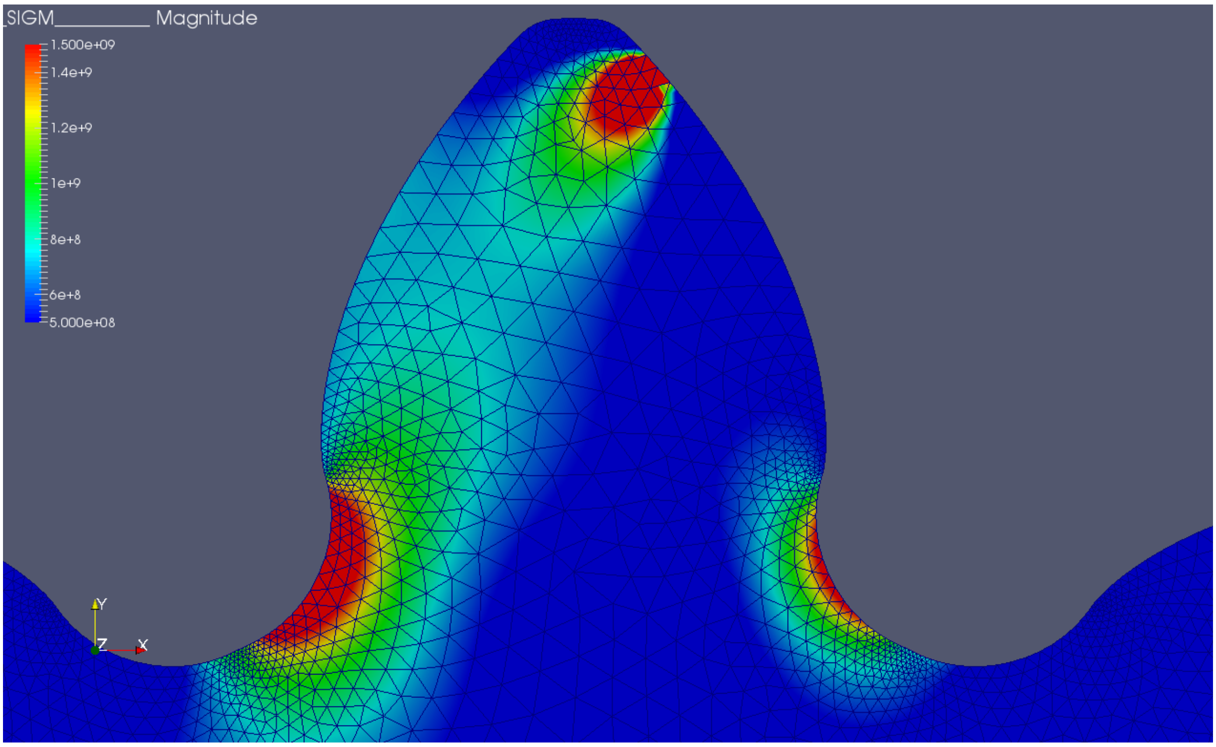


Figure 3.4-2 Tooth stress for case 2. Tensile stress on right flank, compressive stress on left flank. Scale: 500-1500MPa.

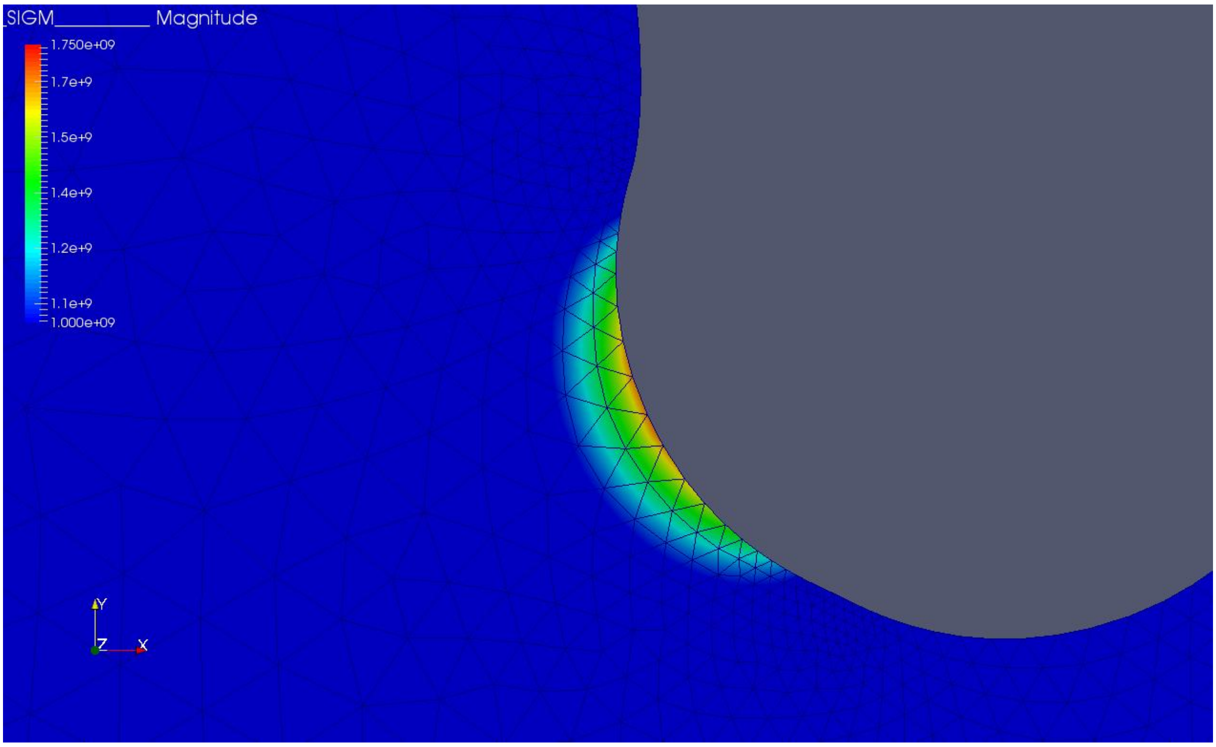


Figure 3.4-3 Tooth stress for case 2. Detail view of root form. Scale is 1000MPa-1750MPa

3.5 Pinion root stress for case 3 (elliptical root shape)

The tooth root shape may be modified to have an elliptical shape to reduce the stress concentration. In the below figure, the orange line is the tooth shape as it results from a generating machining process while the orange line is with an elliptical modification.

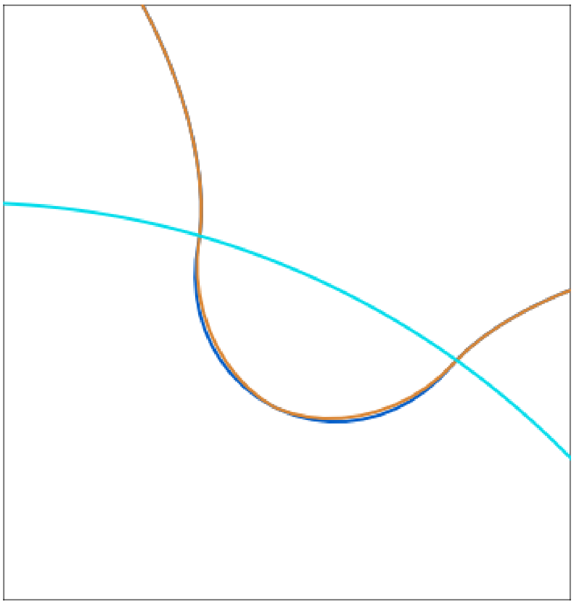


Figure 3.5-1 Elliptical root shape (orange) vs. trochoidal fillet (blue). Elliptical root shape is used in this case 2 calculation.

We then find the below stress distribution. Note that the tooth root stress has dropped to 639.34MPa!

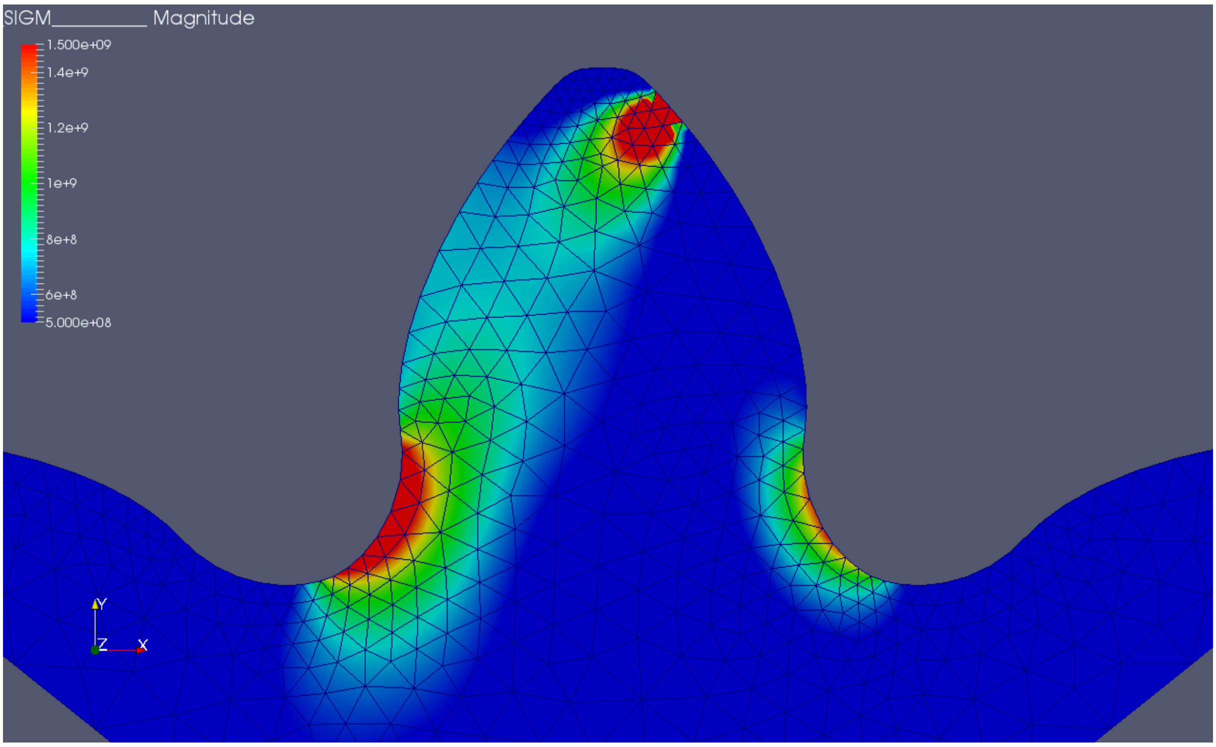


Figure 3.5-2 Tooth stress for case 3. Tensile stress on right flank, compressive stress on left flank. Scale: 500-1500MPa.

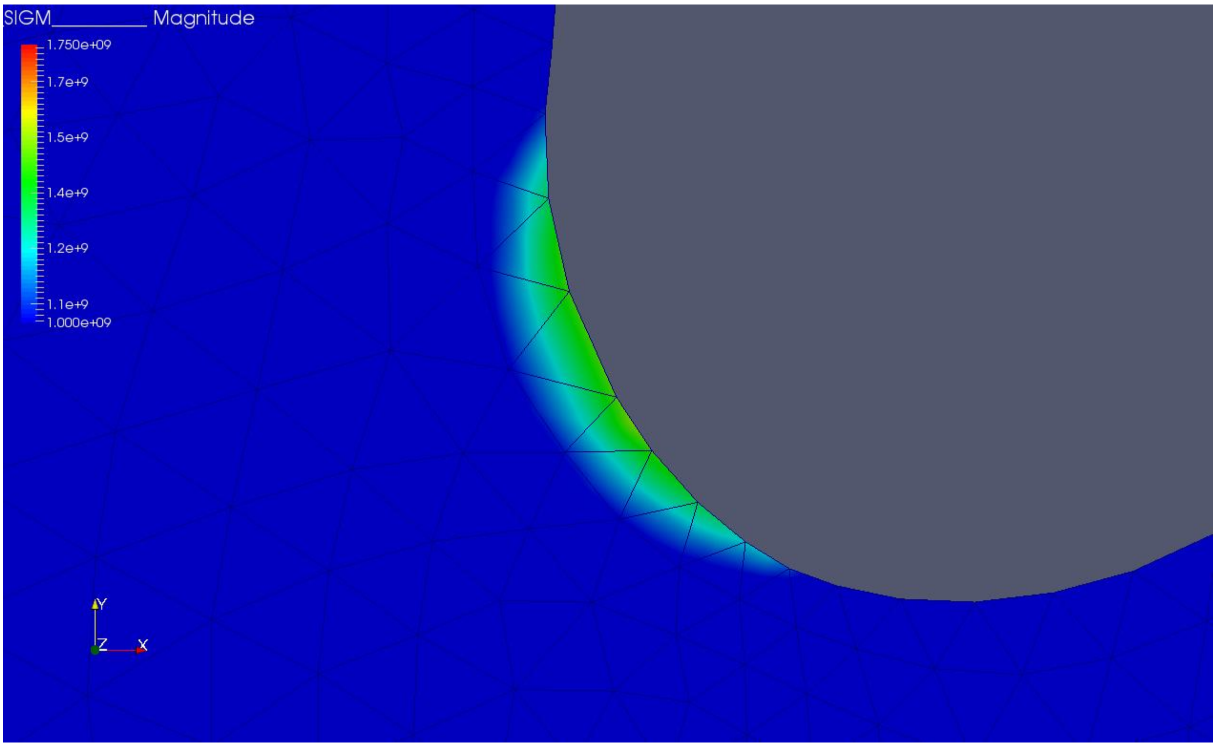


Figure 3.5-3 Tooth stress for case 3. Detail view of root form. Scale is 1000MPa-1750MPa

3.6 Summary

Using KISSsoft release 03-2017, the user can comfortably and accurately compare different tooth root shapes and their effect on tooth root stress. In the above example, a trochoidal fillet, a circular shape and an optimized, elliptical shape are compared. Taking the result from case 1 as the reference, we find the below shown stress levels:

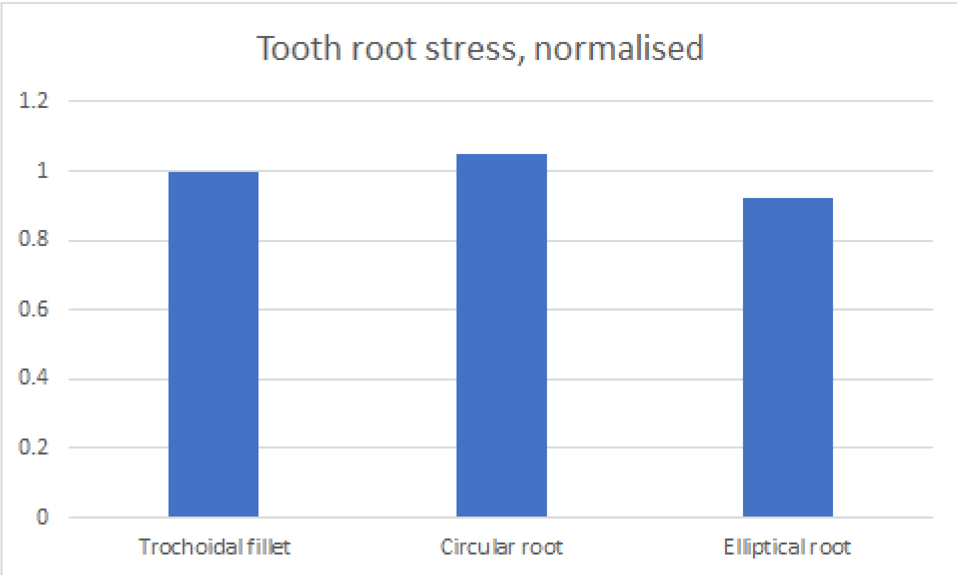
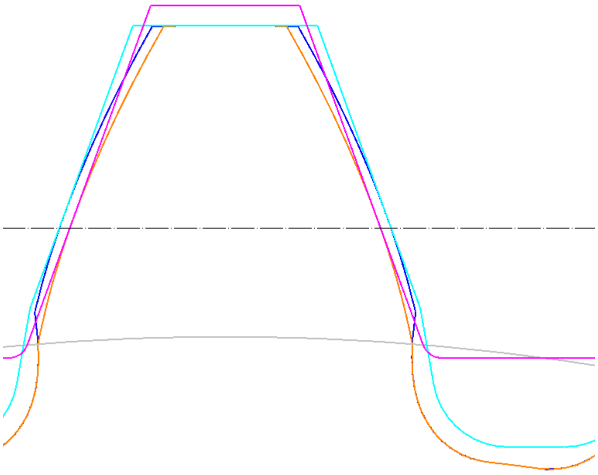


Figure 3.6-1 Tooth root stress level comparison. The root stress may be lowered by about 8% in case of elliptical modification without increase in gear size.

4 Application example, grinding notch

4.1 Gear data with sufficient protuberance

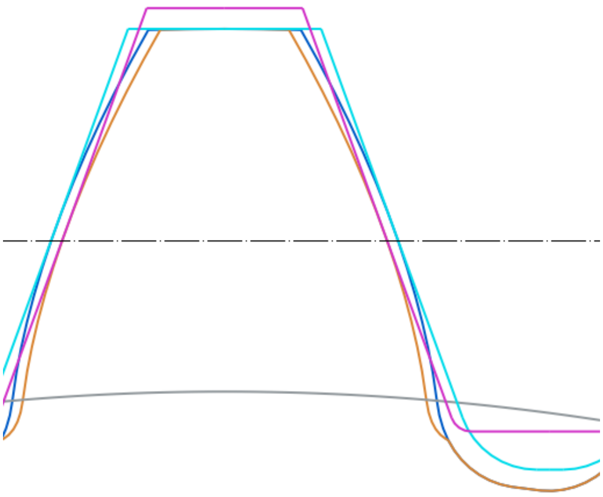
Let us consider a gear with protuberance and a machining stock. The protuberance has been selected such that the machining stock is smaller than the protuberance value. Hence, in the final machining step, only the flank is machined and the resulting root form diameter is lowered.



Blue: gear after hobbing
Orange: gear after final machining
Cyan: protuberance hob basic rack
Magenta: finishing tool basic rack
Grey: dFf after final machining

4.2 Gear data with grinding notch

If the protuberance is absent or not sufficient, a grinding notch results. The grinding notch is machined by the tip of the finish machining tool. If the finishing process is a generating process, then, the fillet is again trochoidal.



Blue: gear after hobbing
Orange: gear after final machining
Cyan: protuberance hob basic rack
Magenta: finishing tool basic rack
Grey: dFf after final machining

4.3 Tooth root FEM of gear with sufficient protuberance

In case of a sufficient protuberance, we now find the highest root stress as shown below.

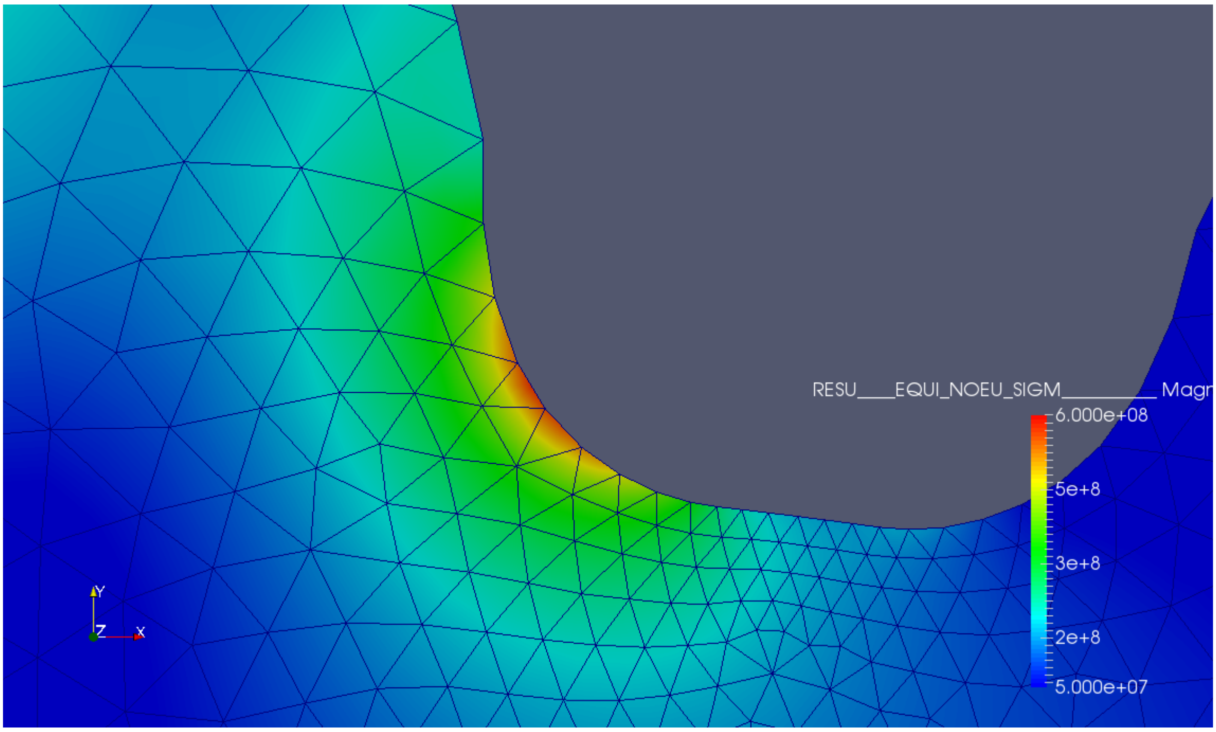


Figure 4.3-1 Stress distribution in root, gear machined with sufficient protuberance.

4.4 Tooth root FEM of gear with grinding notch

In case of a grinding notch, the highest stress is now not in the root itself but in the grinding notch. Obviously, it will increase considerably if e.g. a grinding disk with a lower tip radius is used.

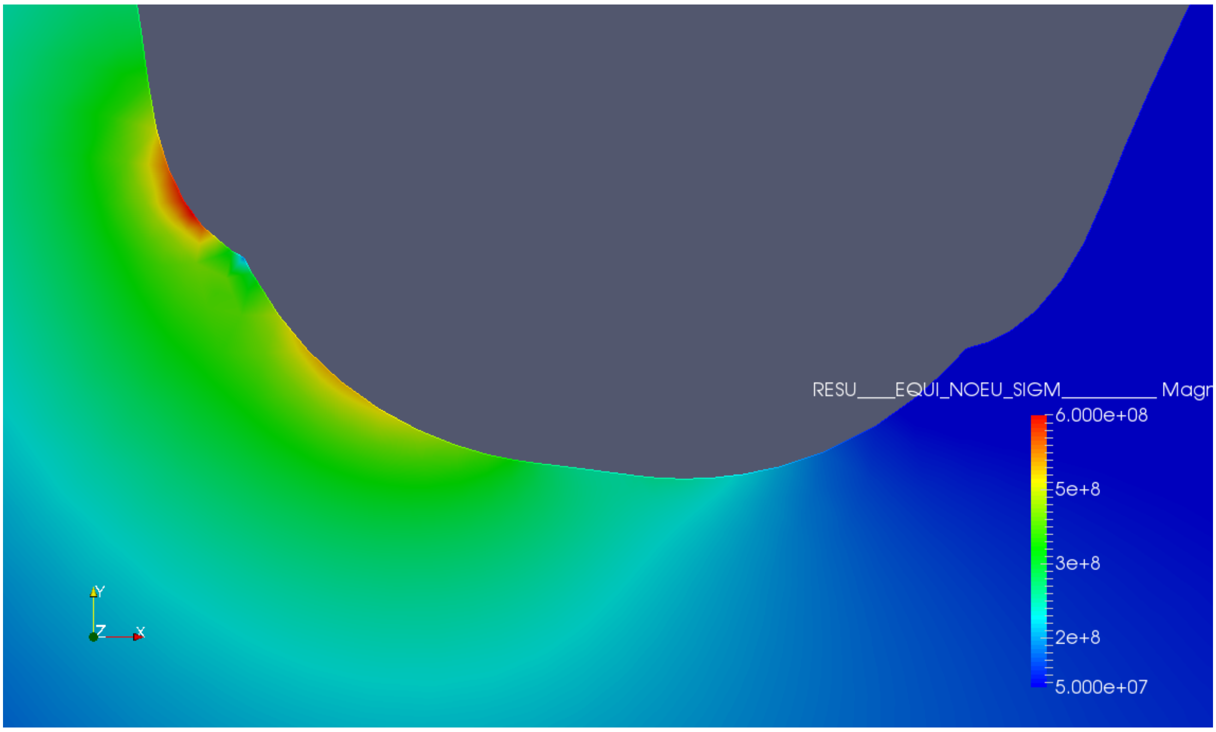


Figure 4.4-1 Stress concentration in the grinding notch.

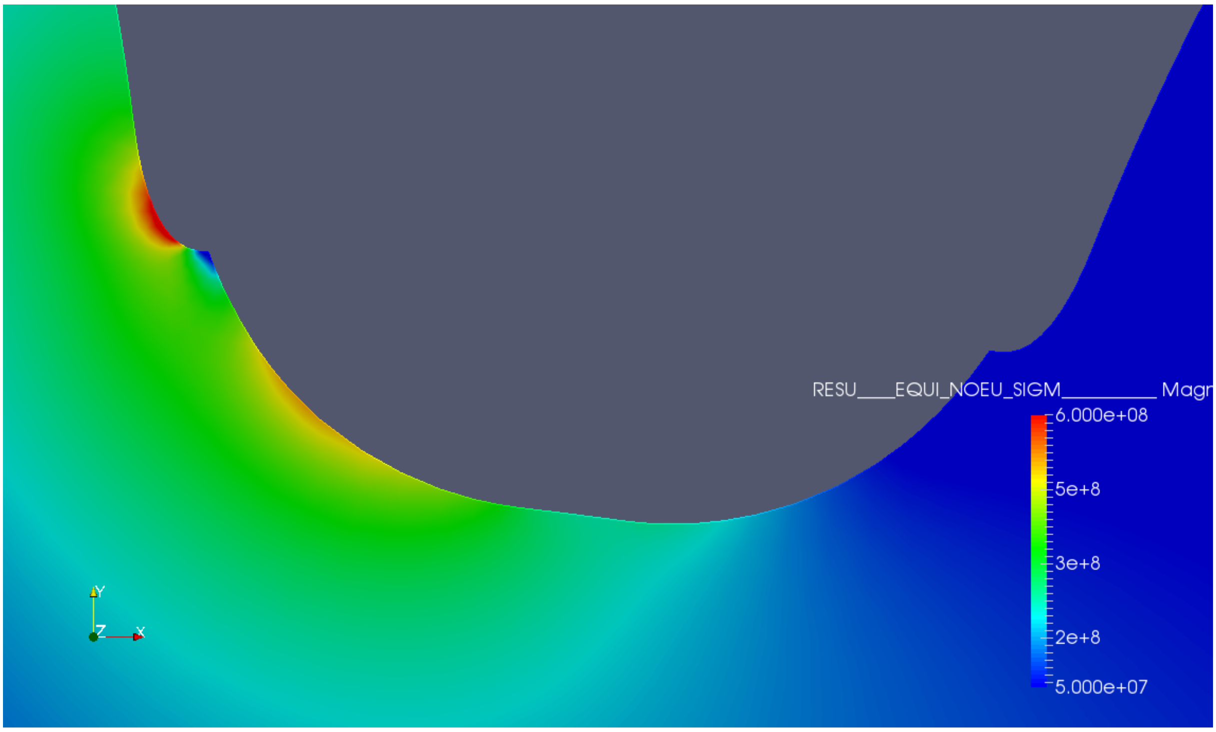


Figure 4.4-2 Increased stress level if the tip radius of the grinding disk is reduced

5 Use of Salome post processor

5.1 Settings in KISSsoft

In KISSsoft, add the calculation “FEM tooth root stress” to your calculation. Then, select that the FEM calculation is done for the pinon. Activate the flag that will start the Salome post-processor after the calculation.

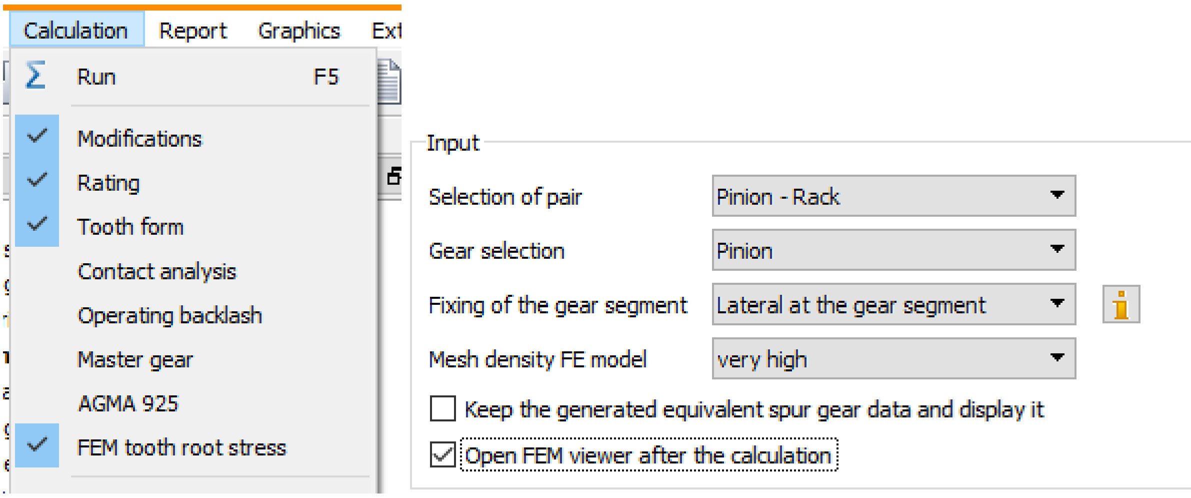


Figure 5.1-1 Settings in KISSsoft. Add the «FEM tooth root stress» calcualtion from the menu «Calculation». Then, select that the FEM calculation is done for the pinion. Select the boundary condition and the mesh density.

Then, run the calculation by pressing “F5”. During the mesh generation and the FEM calculation, the below messages will appear.

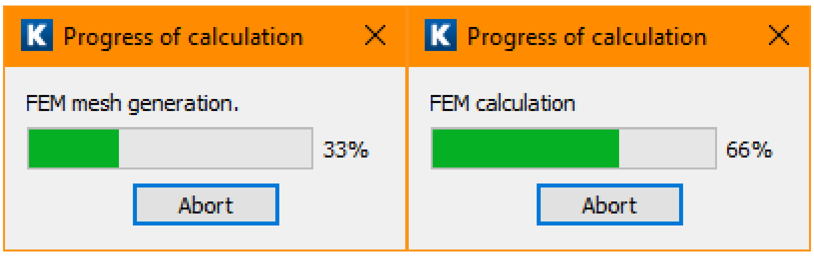
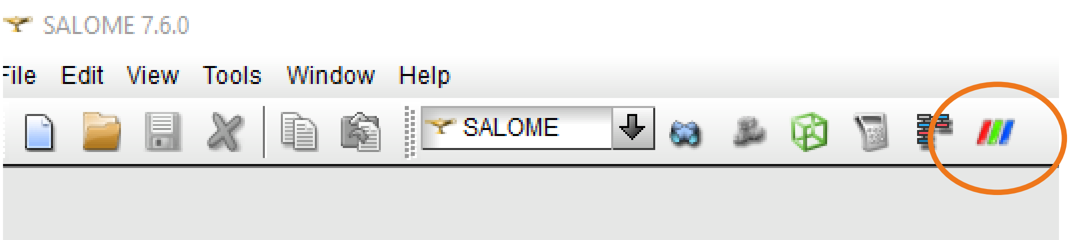
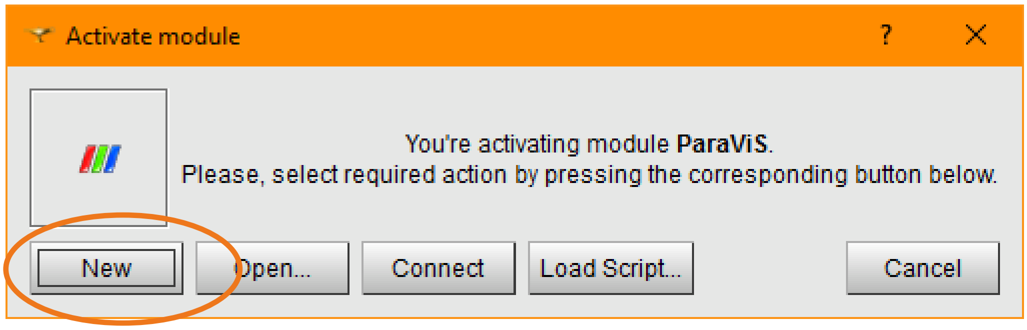


Figure 5.1-2 Messages during mesh generatio and FEM calculation.

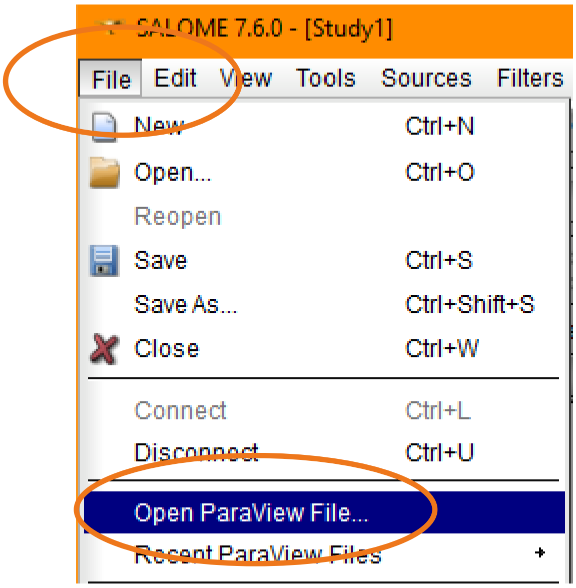
After Salome opens, select “ParaVIS”:



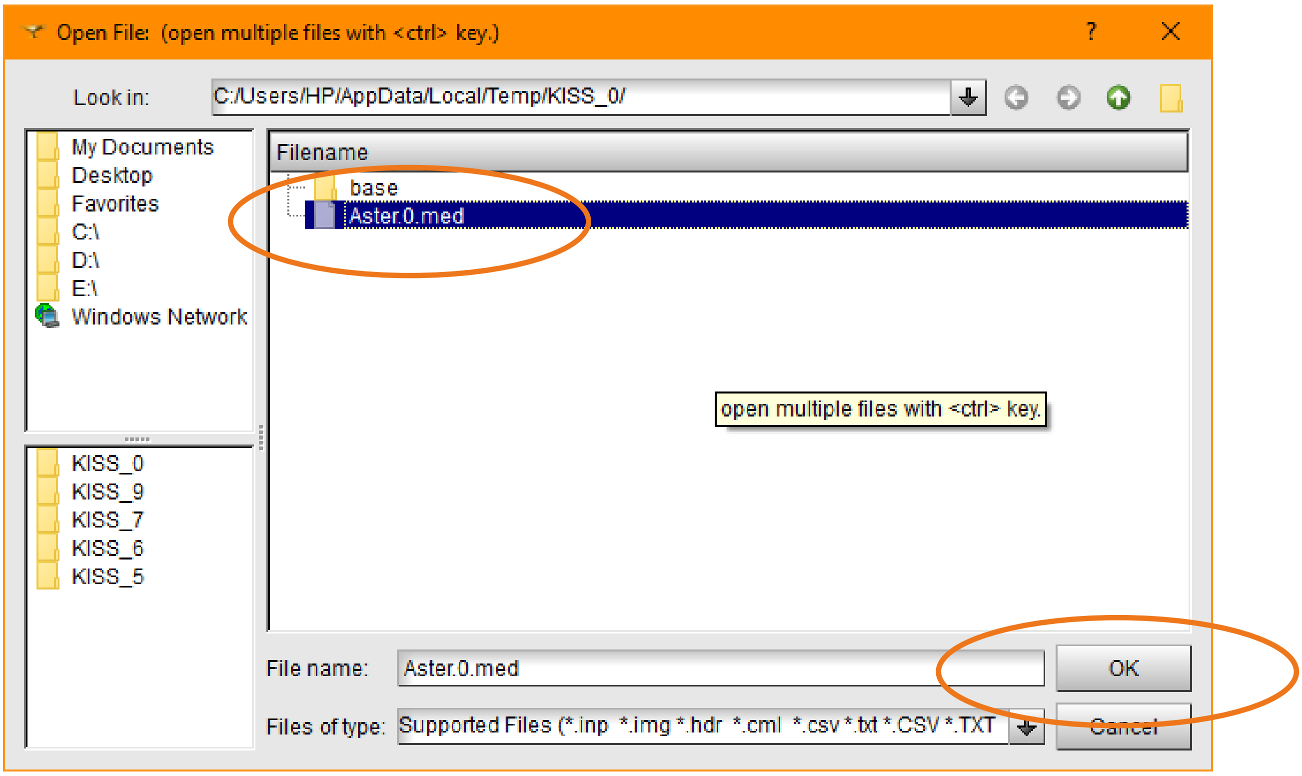
Select “New”



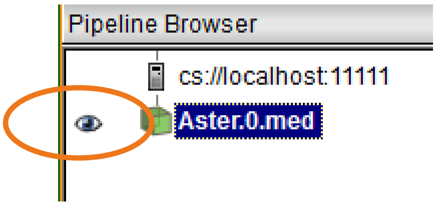
Select “File/Open ParaView File…”



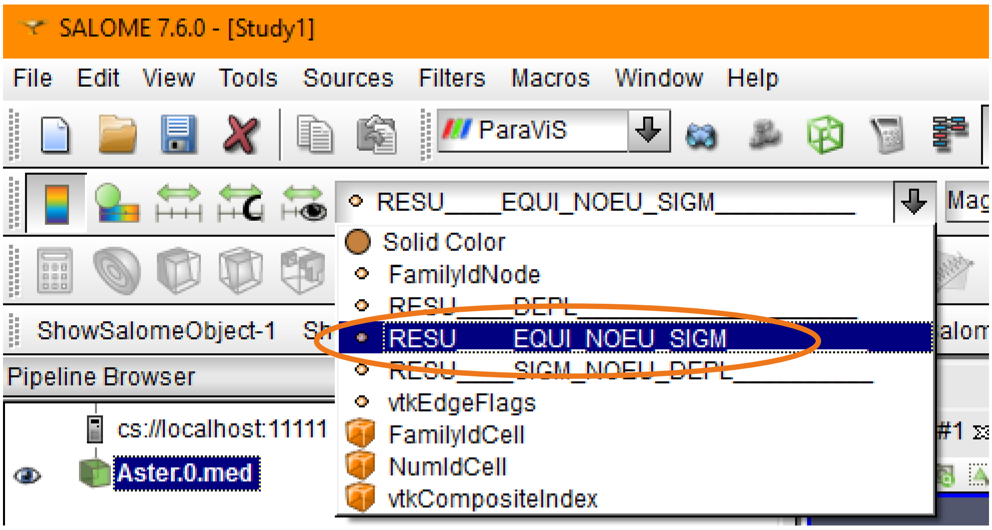
Select the *.med file



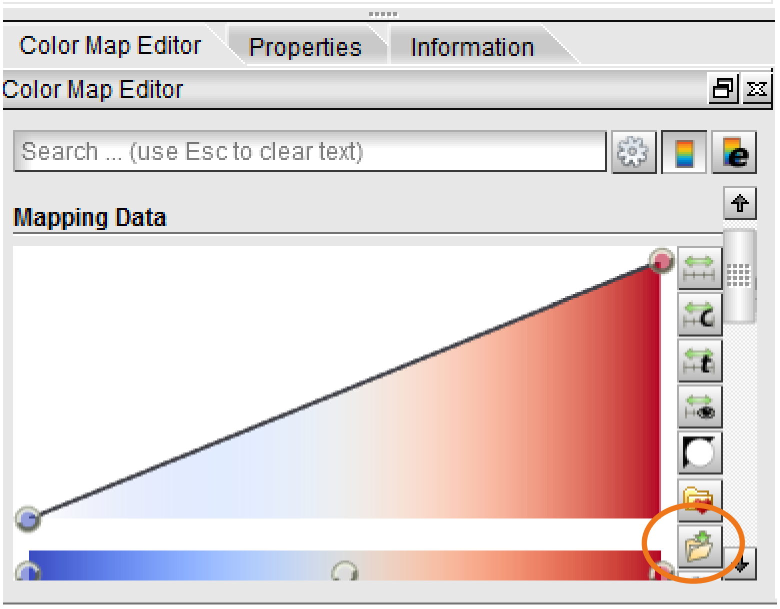
Click on the “eye” symbol:



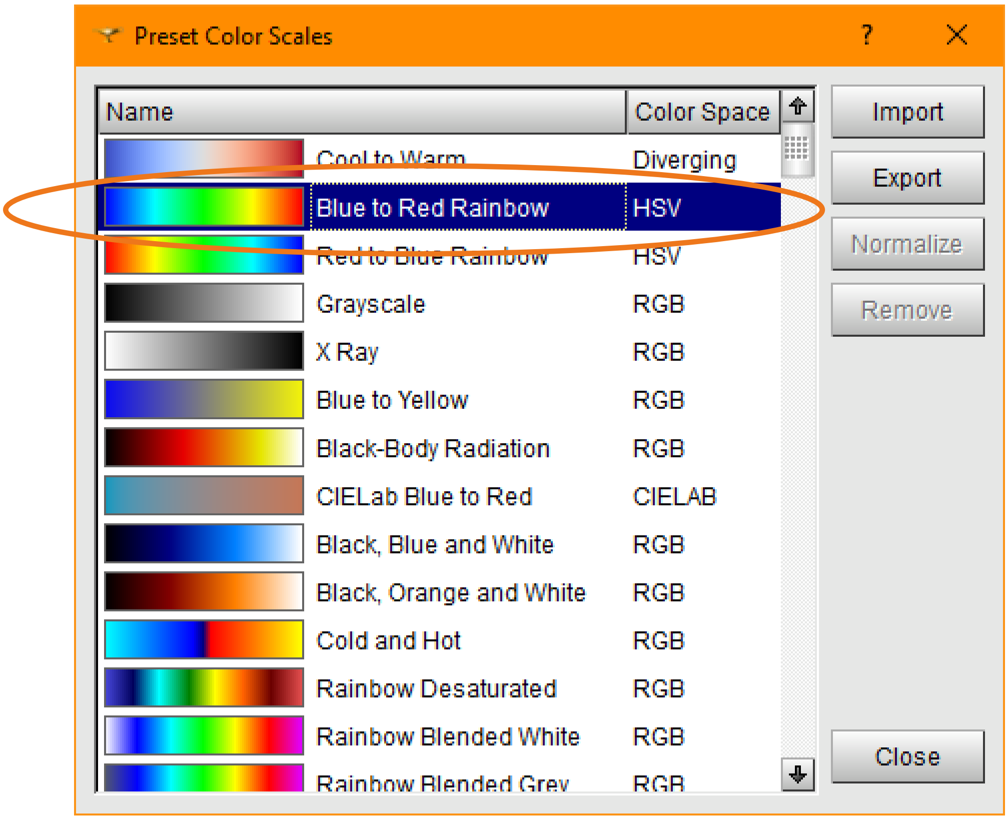
Select that the stress result is shown:



In the color map editor, select the below shown symbol:

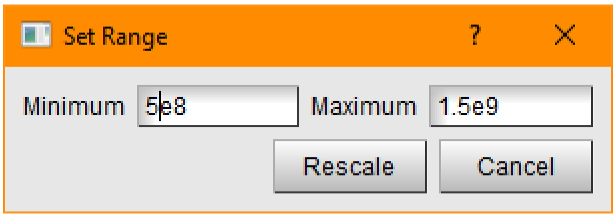


Select a suitable color bar, e.g. and press “Close”

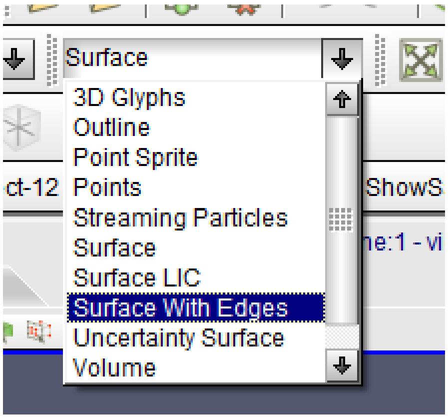


Press the below shown button to change the scale:

Enter a scale (note that units are N/m^2) and press rescale:



Use below selection to see the mesh and the results:



Use the below button to zoom in:

Zoom into the root to get: