Design of Machines and Structures, Vol. 7, No. 1 (2017), pp. 32–39.

MESH INFLUENCE ON CONTACT STRESS RESULTS OF TEETH SPUR GEARS

MARTIN KRATOCHVIL
University of West Bohemia, Department of Machine Design
Univerzitni 2732/8, 301 00 Pilsen, Czech Republic
kratochv@kks.zcu.cz

Abstract: Meshing is very important part of finite element analysis, because it determines parameters of numerical solution of a physics phenomenon. There are many options as type of element, its size, shape or number of nodes. Every physical phenomenon needs different discretization corresponding to equation or set of equations. This paper deals with the influence of mesh settings on results of contact stress of spur gear teeth.

Keywords: *spur gears, contact stress, mesh influence, Ansys*

1. INTRODUCTION

A geometry of gear teeth was created in NX11 CAD system and exported to .stp file. Then the geometry was imported to Ansys Design Modeller. All calculations were done only on one pair of mating teeth (not a complete geometry of a gear). Contact force, geometry and constrains are the same in all cases.

Gear tooth parameters:

- Spur gear, tooth profile according CSN 01 4607 [1]
- Module 6 mm
- Width 20 mm
- Number of teeth 20 (gear 1) and 31 (gear 2)

Material of both gears is Structural Steel from Ansys Engineering Data library. Contact type is Frictionless with Stabilization Damping Factor Value 1.

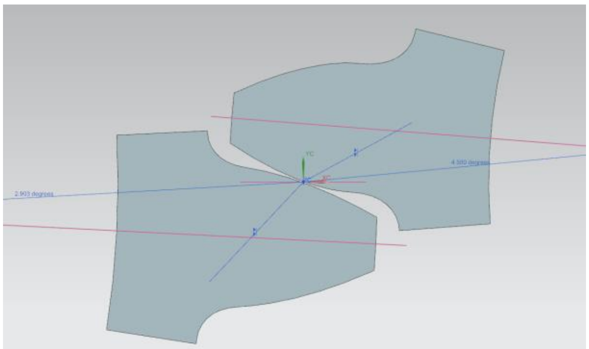

Figure 1. Geometry of teeth (NX11 CAD system)

Meshes with element size lower than 0.5 mm were created on a divided tooth geometry (Model 2). The divided tooth geometry was created in SpaceClaim (a part of Ansys bundle). The aim of divided geometry is to mesh a part of tooth geometry by smaller element size. Smaller element size is used only in small space around contact line. This procedure was chosen to shorten the calculation time. The geometry is preserved, because the nodes of edge elements mate. Mesh connection is set by a function *Form New Part*. A middle geometry is generated as multizone mesh type.

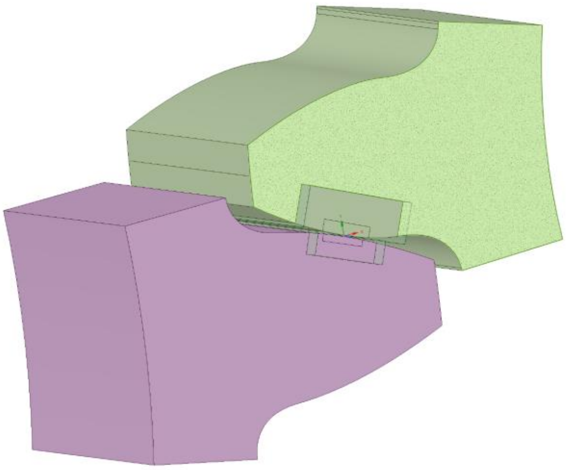

Figure 2. Divided Model 2 geometry in SpaceClaim

Path orientation, boundary conditions and mesh connection are evident from the figures bellow.

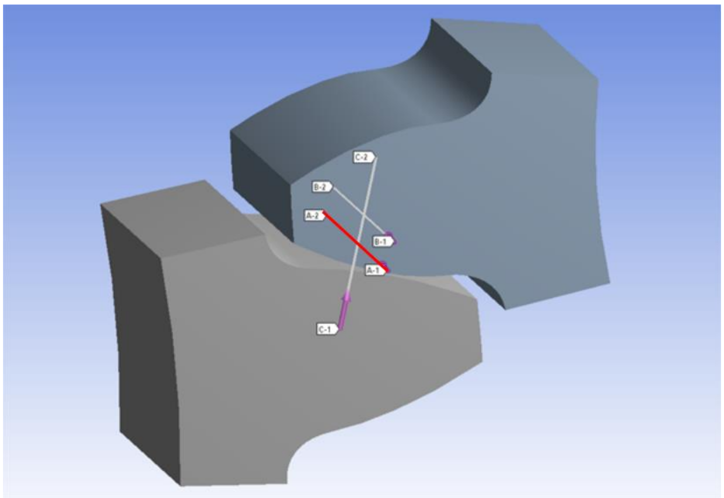

Figure 3. Path-1 orientation (marked by red line)

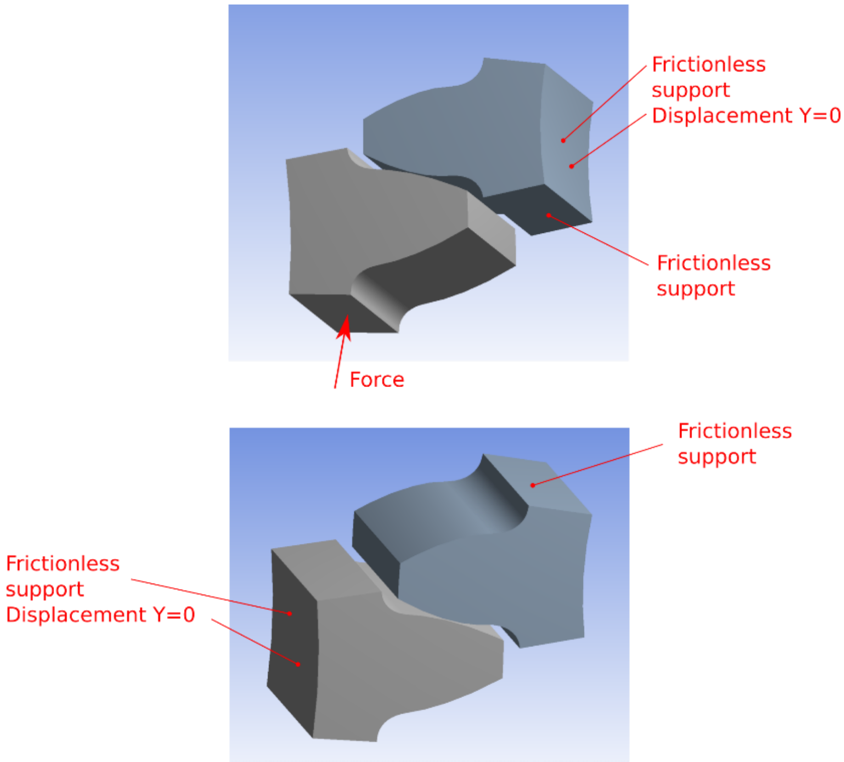

Figure 4. FEM constrains

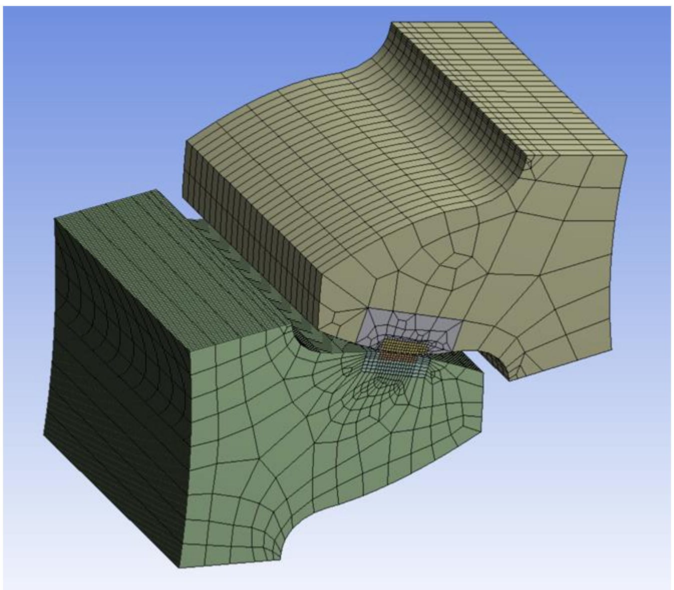

Figure 5. Example of mesh connection of divided volumes (Model 2)

2. EXAMPLES

This part of the paper describes several examples with different mesh settings and its parameters. These options are described in tables. If midside node option isn't specified, then midside nodes are used (kept option).

2.1. Hexahedron vs Tetrahedron, element size estimation (Example 1)

The first calculation was made to determine an optimal size of finite element. Both calculations have been made on simple model with different size of element. The table below describes element type and its size.

Table 1
Example 1 settings

Result	Type of model	Element shape	Element size
tetra 2 mm	Model 1	Tetra	2 mm
tetra 0.5 mm	Model 1	Tetra	0.5 mm
hex 2 mm	Model 1	Hex	2 mm
hex 0.5 mm	Model 1	Hex	0.5 mm

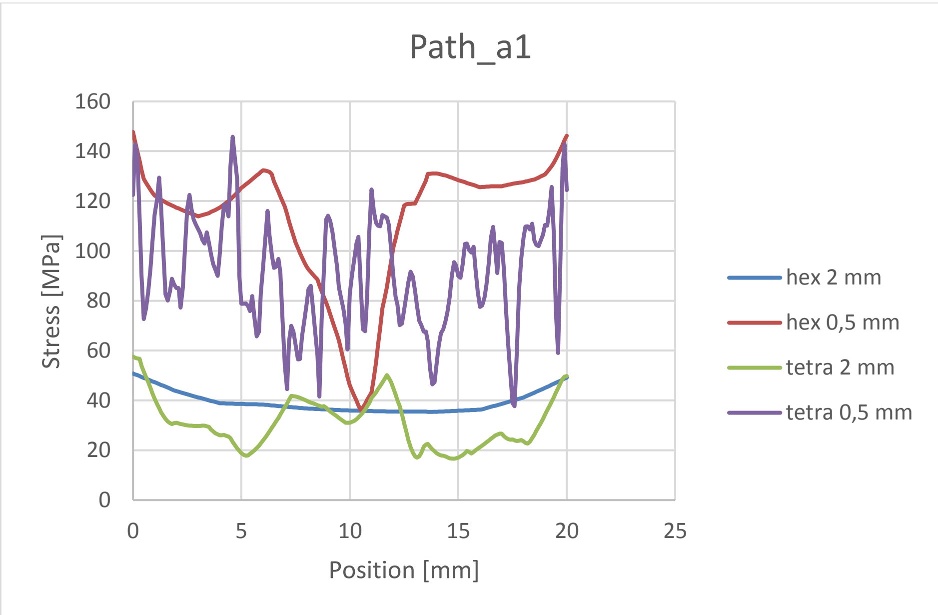

Figure 6. Stress result of Example 1

The graph shows contact stress result on the contact line. It is clear that both element sizes provide improper results. Stress value of 2 mm elements is probably low. It is assumed that smaller elements provide more accurate result. In this case, smaller 0.5 mm tetrahedron elements have higher stress, but large range of values. Values calculated on hexahedron elements are relatively smooth but there is a big gap in the middle of tooth width. After evaluating of theese results, it is necessary to reduce the size of the finite elements around the contact line.

2.2. Hexahedron vs Tetrahedron, suitable element size (Example 2)

The aim of second calculation is to compare 0.5 mm and 0.2 mm size of finite element. Lower element size increases number of total finite element, so it is necessary to optimize the mesh. The mesh optimization is based on the Model 2, which consists of three divided volumes. Volumes in contact have 0.2 mm element size.

Table 2
Example 2 settings

Result	Type of model	Element shape	Element size
tetra 0.5 mm	Model 1	Tetra	0.5 mm
hex 0.5 mm	Model 1	Hex	0.5 mm
tetra 0.2 mm	Model 2	Tetra	0.2 mm
hex 0.2 mm	Model 2	Hex	0.2 mm

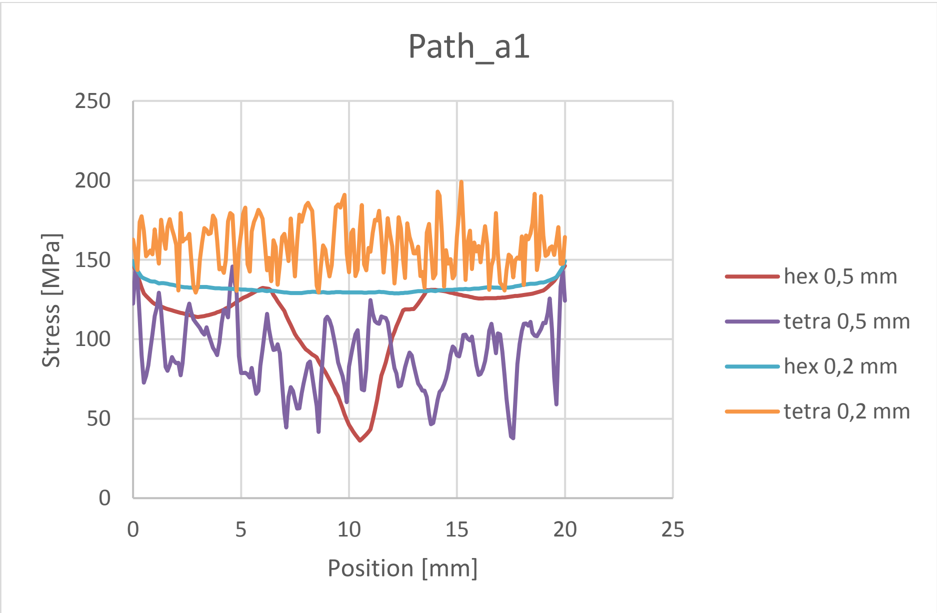

Figure 7. Stress result of Example 2

The element size of 0.2 mm seems to provide better result than 0.5 mm or 2 mm. The difference between higher and lower stress of result of tetrahedron elements is smaller than 0.5 mm element size. In the case of 0.2 mm hexahedron element, the gap mentioned in previous analysis disappeared. The stress calculated on hexahedron element is evenly distributed, which corresponds to a static load and a precisely made tooth (nominal dimensions without tolerances). Tetrahedron elements have still relative high range of values, but average contact stress as a result from tetrahedron mesh could be calculated.

2.3. Influence of midsize nodes (Example 3)

This calculation explains the influence of midside nodes. Midside nodes have certain importance in some cases, where use of linear elements requires a much larger number of elements to get appropriate result. There is the same element size to demonstrate the difference.

Table 3
Example 3 settings

Result	Midsize	Type of model	Element shape	Element size
tetra 0.2 mm dropped	Dropped	Model 2	Tetra	0.2 mm
hex 0.2 mm dropped	Dropped	Model 2	Hex	0.2 mm
tetra 0.2 mm	Kept	Model 2	Tetra	0.2 mm
hex 0.2 mm	Kept	Model 2	Hex	0.2 mm

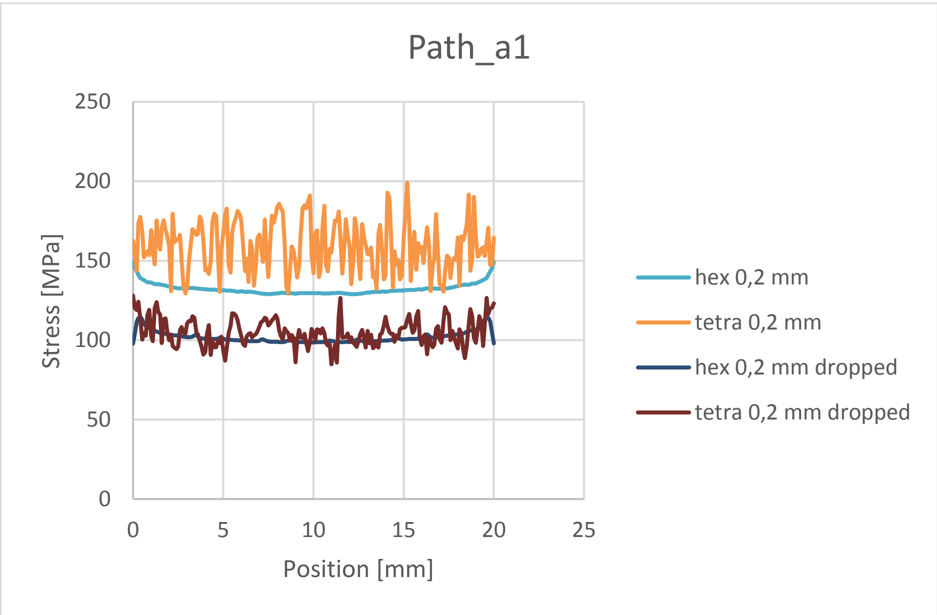

Figure 8. Stress result of Example 3

Stress calculated on both linear hexahedron and linear tetrahedron elements is lower than values calculated on elements with midside nodes. Average values of contact stress on linear elements are almost identical. But the values of contact stress are relatively low, smaller element size of linear elements should be better, but lower size increases number of elements and calculation time. For this reason, the use of linear elements is not appropriate.

2.4. Inflation meshing (Example 4)

Inflation is a method of meshing based on creation of layers, a layer thickness obviously graduates with growing distance from an influence area. Ansys can create inflation based mesh that consists of linear elements (dropped). For comparison, a mesh of hexahedron elements with 0.08 mm (Model 2) size was created.

Table 4
Example 4 settings

Result	Midsize	Type of model	Element shape	Element size
hex 0.2 mm	Kept	Model 2	Hex	0.2 mm
Inflation	Dropped	Model 1	Hex	0.2 mm
Hex 0.08 mm	Kept	Model 2	Hex	0.08 mm

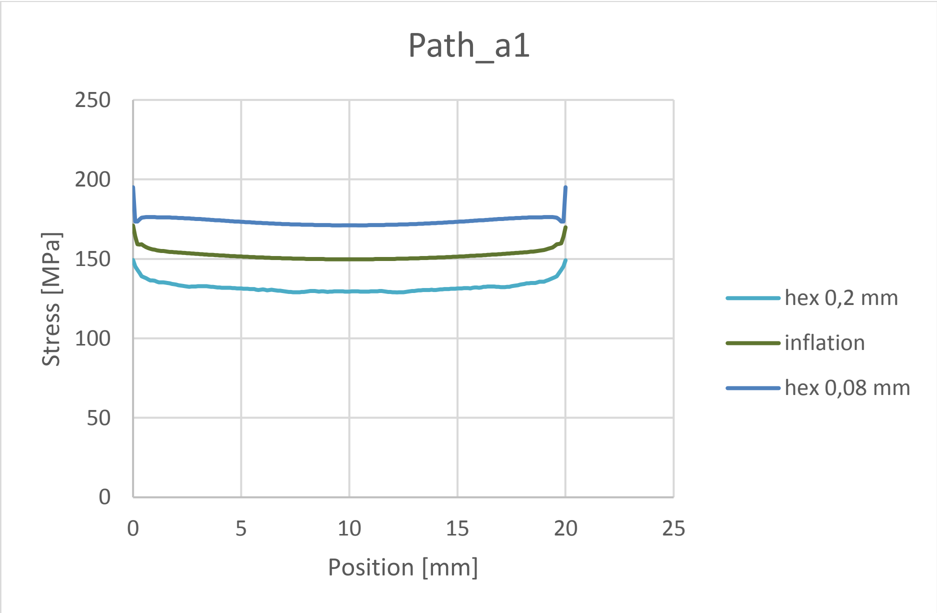

Figure 9. Stress result of Example 4

Lowering the size of element to 0.08 mm increases contact stress. This result is probably more accurate than the mesh with 0.2 mm element size. But theoretically, 0.08 mm size mesh has approximately 15 times more elements than 0.2 mm. In this case, divided volume (Model 2) decreases total number of elements because 0.08 mm size elements are only close to the contact line (2,2 million nodes, 520 000 elements, elapsed time 47 000 seconds). Calculation of inflated mesh was significantly faster (elapsed time 5930 s, 730 005 nodes, 172 500 elements). The fastest was 0.2 mm hexahedron mesh (elapsed time 1174 s, 244 273 nodes, 64 520 elements). Elapsed time of all calculations can be different on another computer. Inflated mesh seems to be perspective in the case of teeth contact, but further verification is needed.

3. Summary

Results of examples previously described suggest, that too large elements cannot provide proper values of contact stress. Smaller elements with size 0.5 mm lead to more appropriate values, but the optimum is probably 0.2 mm element size or lower. These sizes provide higher stress values and lower range of maximal and minimal contact stress. Calculation with linear elements of the same element size (as dropped) is faster than quadratic elements, but the contact stress is too low. The results suggest that use of linear elements in the case of inflated mesh could make sense, because there is higher stress probably caused by higher number of layers.

4. Acknowledgement

The research work presented in the paper was supported by a project SGS-2016-012.

5. References

[1] CSN 01 4607. *Gears with involute profile: Basic Profile*. Prague, Office for standardization and measurement, 1978 (in Czech).