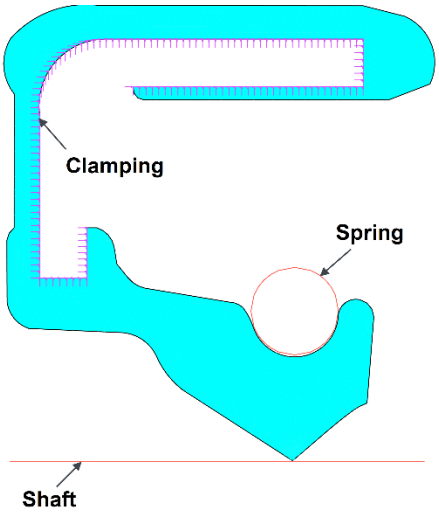
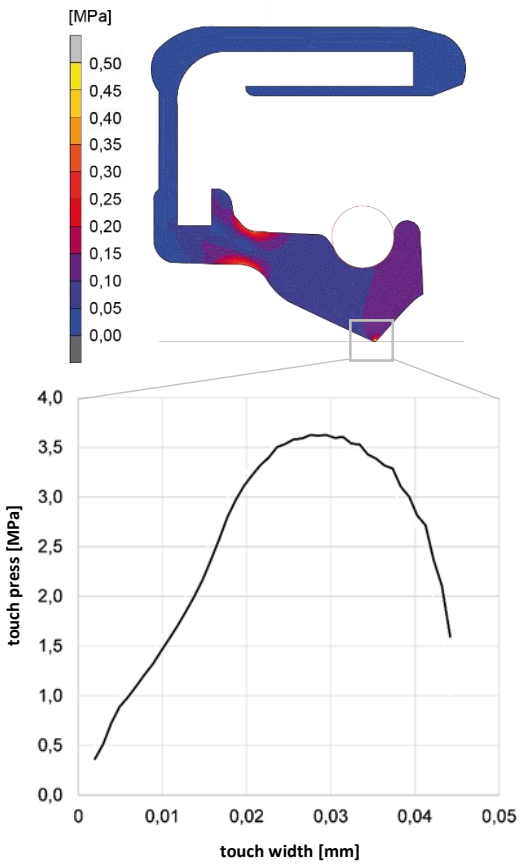
Finite element simulation

IMA-TechSheet #103010 V2

University of Stuttgart
Institute of Machine Components



Finite element model of a shaft seal



Equivalent stress and contact pressure

Description:

- Geometry model from a profile section
- Networking MSC Patran (2D) or Altair HyperMesh (3D)
- So-to-say-static 2D or 3D assembly simulation of a sealing element in a groove or on a shaft/rod using the FEM program MSC Marc Mentat
- Shaft, groove or rod usually ideally rigid
- Definition of displacement and contact boundary conditions
- Definition of a suitable material model (usually non-linear for rubber-elastic materials or thermoplastic materials such as PTFE)
- Definition of load cases, e.g. pressure, temperature, etc.
- Assignment of a suitable element type

Available Test Methods:

- Analysis of general structural mechanical quantities such as stress and deformation
- Investigation of contact pressure and contact geometry
- Geometry optimisation on the basis of the above-mentioned result evaluation