Federal State Autonomous
educational institution
Higher Professional Education
"Siberian Federal University"
Polytechnic Institute
Department "Design and technological support
Machine-Building Industry"

Optimization design of molds data for the manufacture of plastics and automation of its designing

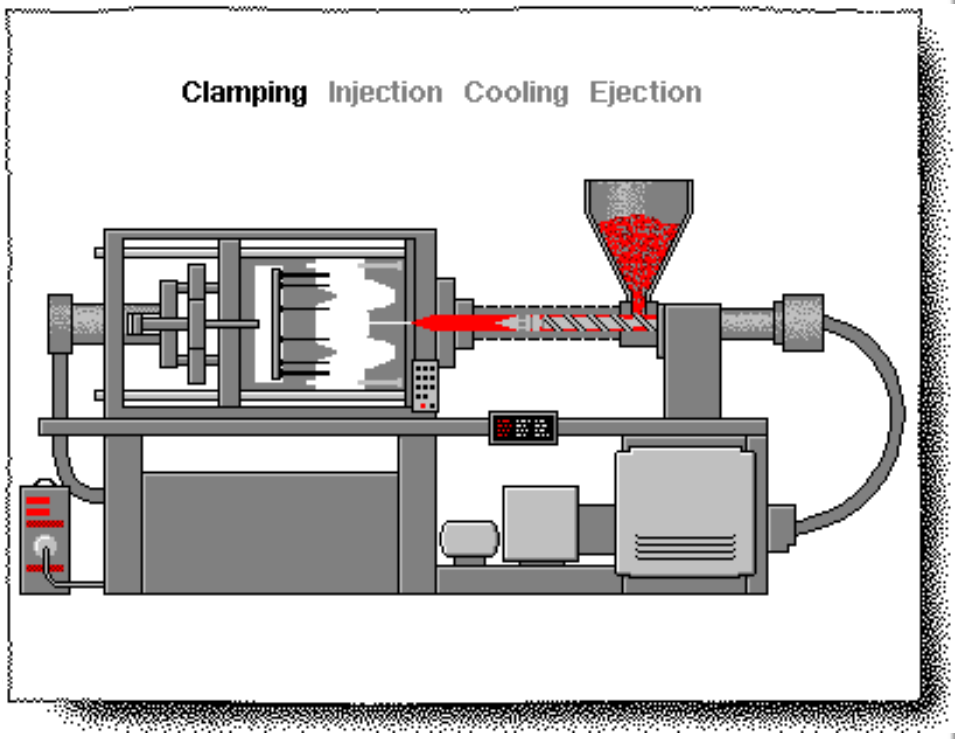
Student of the 2nd course of Master degree: A.S. Demyanenko

Scientific Supervisor: M. P. Golovon, PhD in Engineering

СИБИРСКИЙ ФЕДЕРАЛЬНЫЙ УНИВЕРСИТЕТ
SIBERIAN FEDERAL UNIVERSITY

The objectives of the study are

1) Development and verification of a mathematical model of the process material in molding products of complicated contour profile;
2) Analysis and calculation of temperature fields in the area of formation taking into account the constructive elements of the performance of heating and cooling channels;
3) Sealing process optimization due to the configuration changes and local changes in forming the surface and cross-sectional area of the feeding channels (sprues).



Internship Plan

	Events	Result	
1	Learning techniques for solving engineering problems in the ANSYS environment.	Skills to solve engineering problems in ANSYS were obtained .	✔
2	The study of methods and technologies solve multiphysics problems in the ANSYS environment.	The study of combination of different types of analysis.	✔
3	Study of parallelization of computational processes in dealing with complex tasks in ANSYS environment.	The Study of parallelization of the task to speed up its decision.	✔
4	Analysis of possibilities for automation of thermal calculations, the process of filling the formative elements of the cooling mold melt in ANSYS environment.	The study of thermal analysis and analysis of the results. ACT Console for the Study of the concept basic commands and drawing the simplest algorithms for Ansys Workbench software	⏳
5	The study of problem solving with different contacts.	Study of contact analysis is useful as there are a lot of contact surfaces in my thesis.	⏳
6	Visit the Munich Technical University and other leading research centers in Germany. Introduction to modern scientific and technical developments.	Unfortunately to visit Technical University of Munich wasn't possible.	✘

The content of the first week

ANSYS Workbench / Software handling

- Introduction to FEM
- Demonstrator (live)
- Workbench Project page
- Material Definition
- Objects and their properties
- Coordinate Systems
- Mechanical software handling
- Named Selection Worksheet

Discretization / Theory

- Meshing (Theoretical Introduction)
- Element size of thin Structures
- Geometry Preparation
- Global Mesh Settings
- Local Mesh Settings
- Mesh based simplification
- Connecting bodies

Boundary Conditions / FE Idealization

- Introduction to Boundary Conditions
- Deformation-Boundary Conditions
- Remote Points
- Nodal Coordinate Systems
- Introduction to Nonlinear Statics
- Load-Boundary Conditions
- Inertial Loads
- Nonlinear Boundary Conditions-Contact

Evaluation of Results

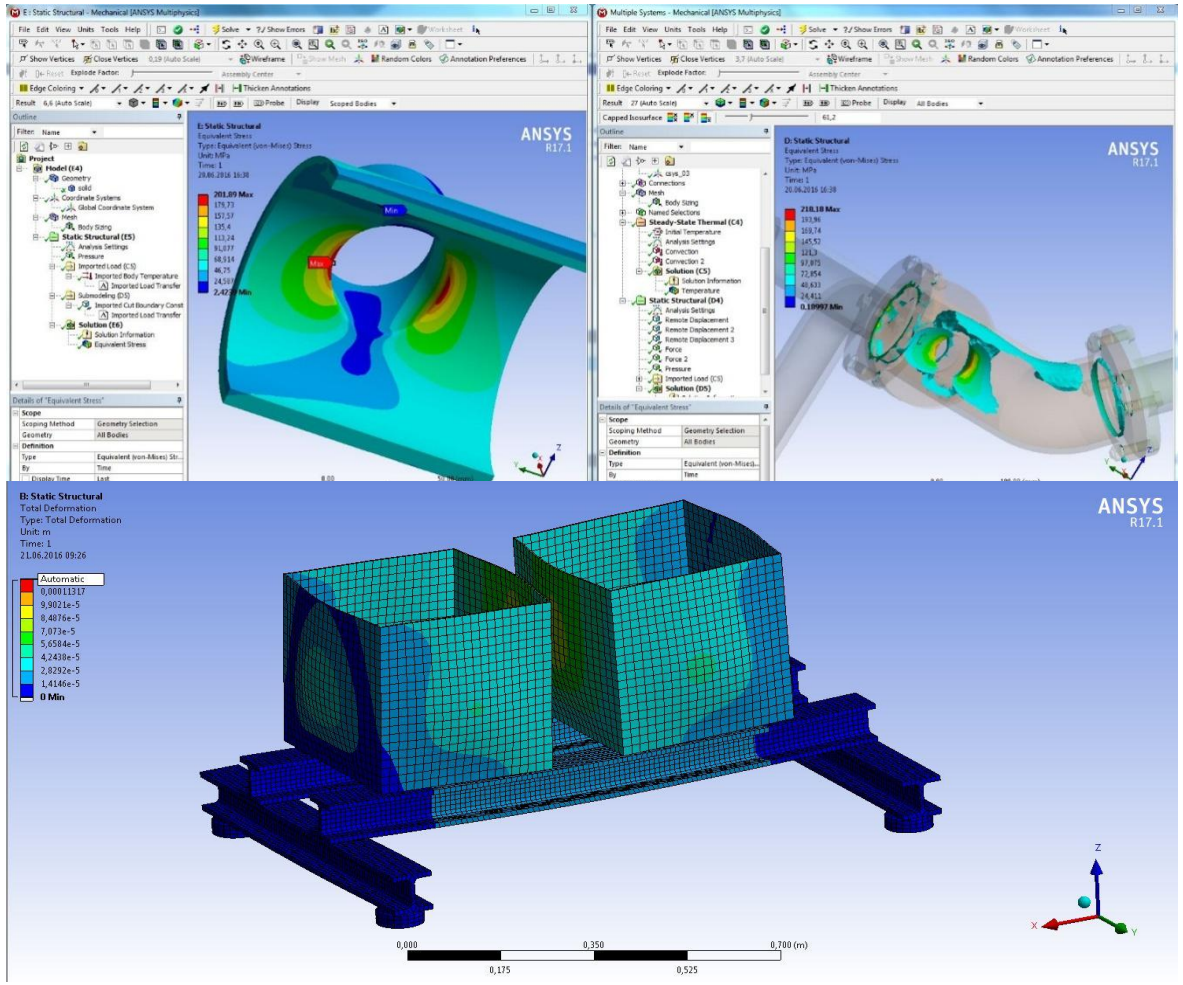
- Evaluation of Results
- Adaptive Mesh Refinement
- Singularities
- Evaluation in Cylindrical Coordinates
- Construction Geometry - Path Evaluation
- Probes
- Submodeling
- Computation of large Models (HPC)

СИБИРСКИЙ ФЕДЕРАЛЬНЫЙ УНИВЕРСИТЕТ
SIBERIAN FEDERAL UNIVERSITY

ANSYS Mechanical – Software Handing

A 400°C hot gas flows in a flange which is supposed to be examined in terms of developing maximum stresses with a thermal-structural coupled analysis.

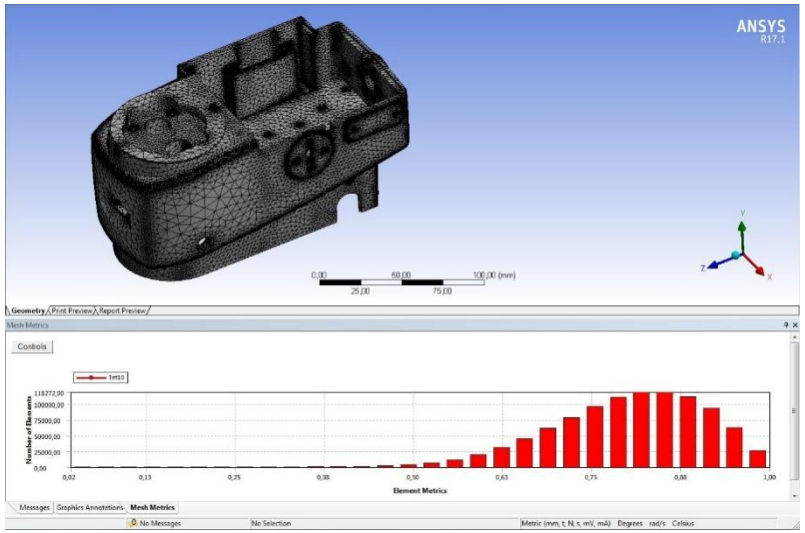
I have learnt how to work with a temperature analysis that is useful for the verification data of the mathematical model in my thesis.



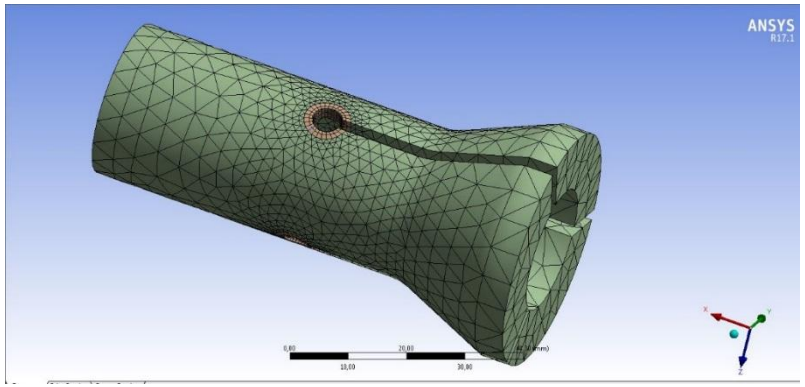
Mesh of finite element method

In this chapter we have studied the types of elements, Shape and Solid functions, integration points as well as the conclusion of the equilibrium of the system Ku = F which is one of the following key issues. The choice of the element size, the setting of the global and the local mesh and the preliminary mesh settings geometry were studied.

In this picture you can see globally and locally customized mesh condition.



One of the most important stages of my thesis is to create a calculation model. Mesh is the important stage for obtaining real results.



Topology Optimization

- **Material along the load paths**
- Motivation
- Concept of the topology optimization
- ACT Extension
- 2D Michell-structure (Hands-on)

- **Without restriction it will not work**
- Design constraints
- Manufacturing constraints
- Generic engine mount (Hands-on)

- **Different ways to get the optimal design**
- Objective functions
- Comparison of different objectives (Hands-on)
- Single Compliance vs Multiple Compliance (Hands-on)

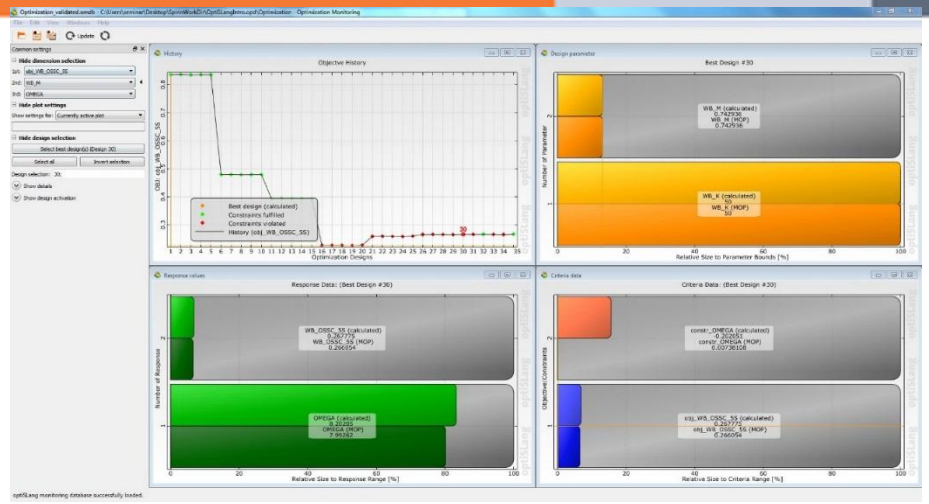
- **Redesigning**
- ANSYS Topology Optimization – ANSYS SpaceClaim (Hand-on)

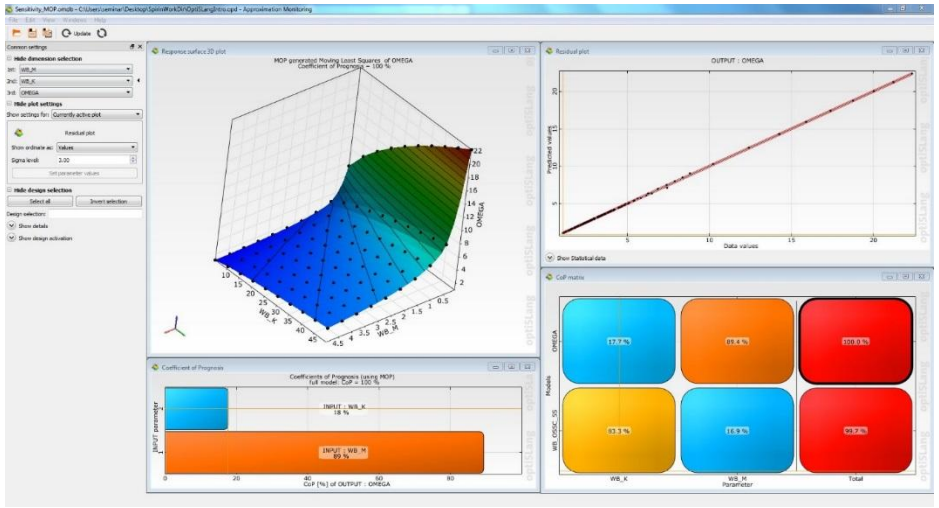
ANSYS Topology Optimization

The aim of optimization of the topology:

To get a material distribution which provides for a given design space and for a single or multiple load case scenario of an optimal part stiffness.

The volume of the part is usually definited as constraint. The design variable is the pseudo density which is assigned to each element. Value „1“ describes that the element is active. „0“ means inactive





The content of the second week

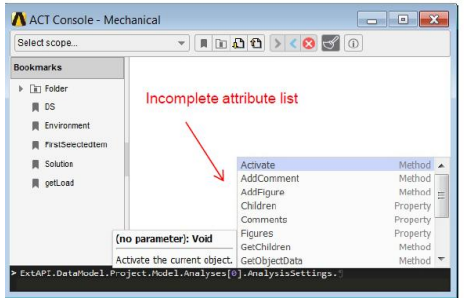
- **First day**
- Introduction ACT
- (Iron) Python
- XML format
- Toolbar
- Journaling (project schematic)
- **Second day**
- ACT console
- Change and insert standard feature
- Pre-processing feature (reuse APDL)
- Post processing feature
- Graphic
- Create report
- **Third day**
- Exercise: fix displacement
- Compiling an extension
- Wizard
- Optional topics
  * DesignModeler
  * Insert meshfeature
  * Rename by class
  * Debugging_with Visual Studio

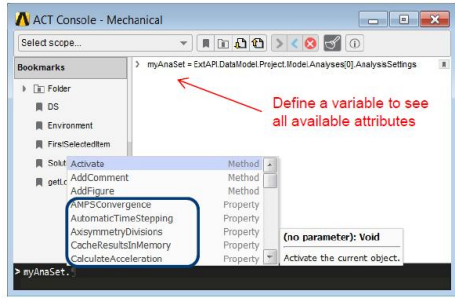
ACT Console

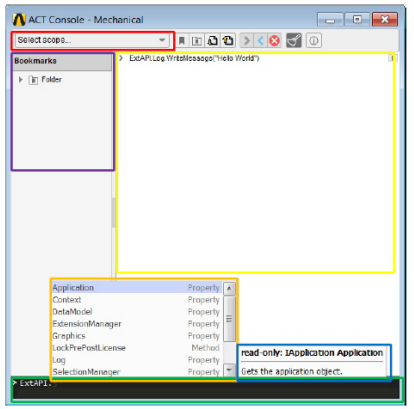
ACT extension

- An ACT-Extension is the complete Application with the integration into Workbench.
- An ACT-Extension is based on the Language XML and Python.
- An ACT-Extension can be:
- In-house solver integration
- New post-processing features
- Custom results
- Workbench access
- GUI integration
- Toolbars
- Graphics
- Result visualization

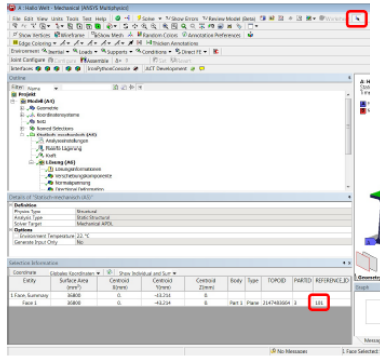
The small amount of time allotted was unable to detail and fully given material.

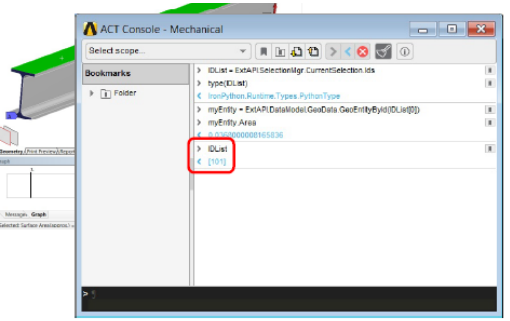






1. **Context menu:**
   - In which environment you want to test
2. **Command line:**
   - To drop commands
3. **Method/Property list**
   - Available methods and properties of the object
4. **Information Window**
   - e.g.: which transfer parameters are
   - Required
5. **Command History Window**
6. **Bookmarks**
   - to bookmark predefined commands for later insertion

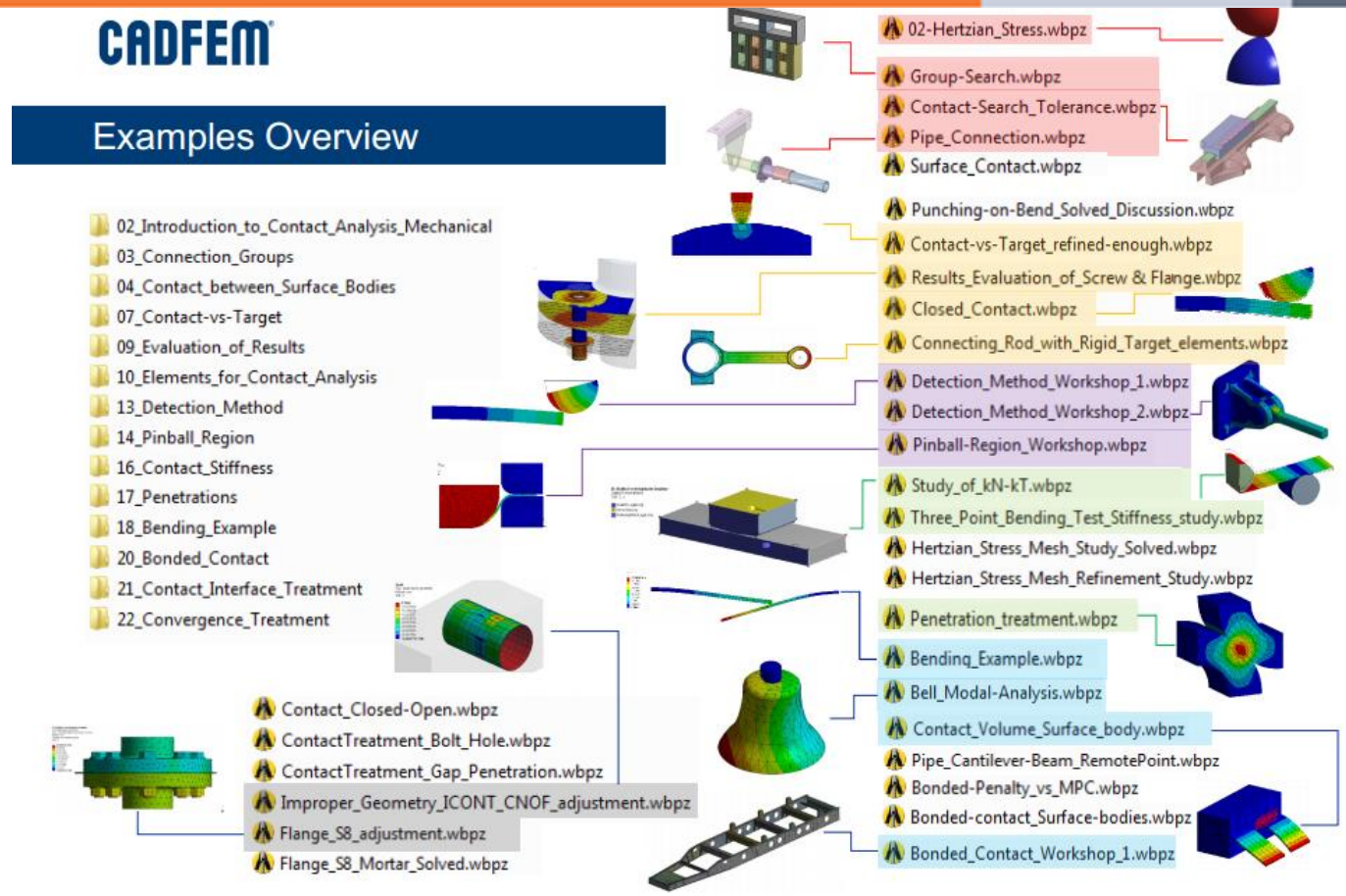




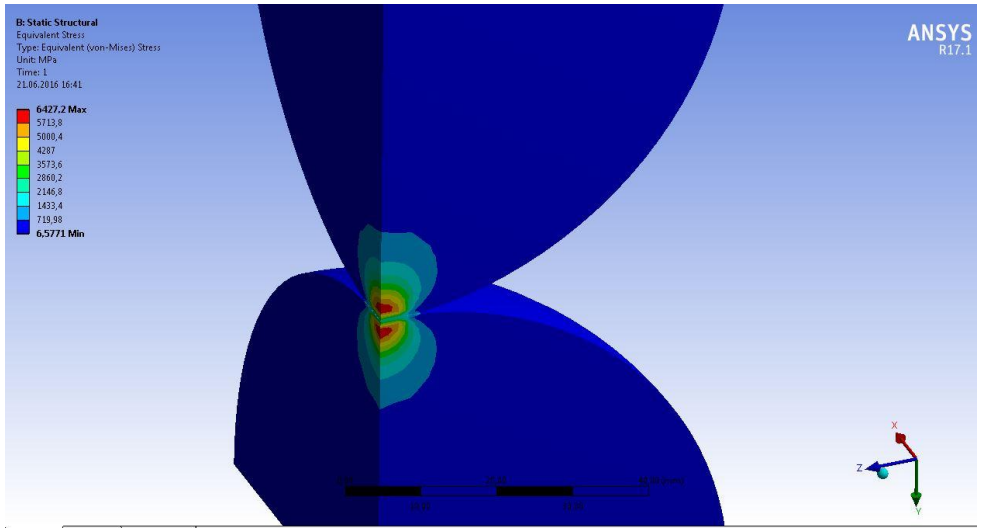
СИБИРСКИЙ ФЕДЕРАЛЬНЫЙ УНИВЕРСИТЕТ
SIBERIAN FEDERAL UNIVERSITY

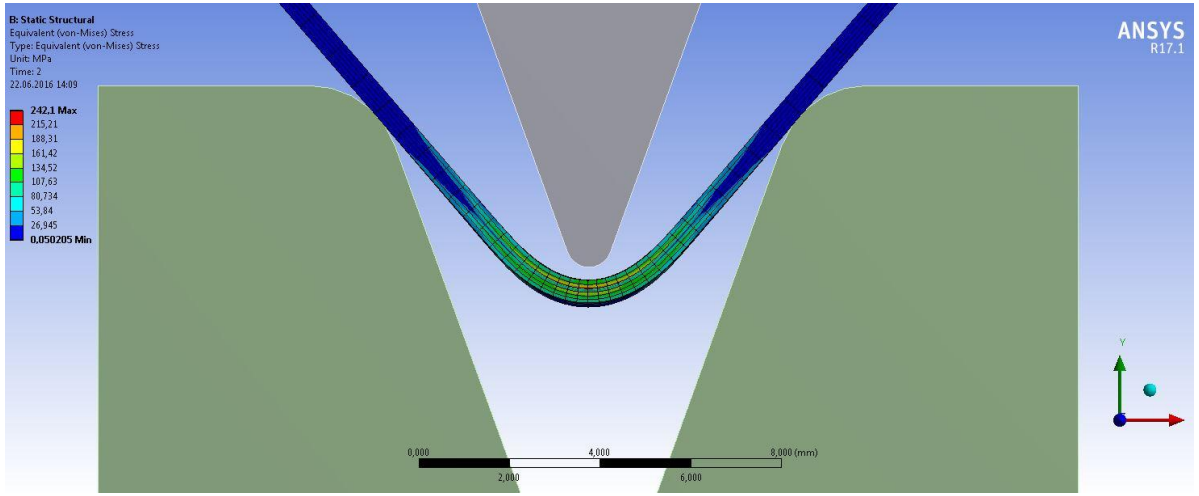
The content of the final week

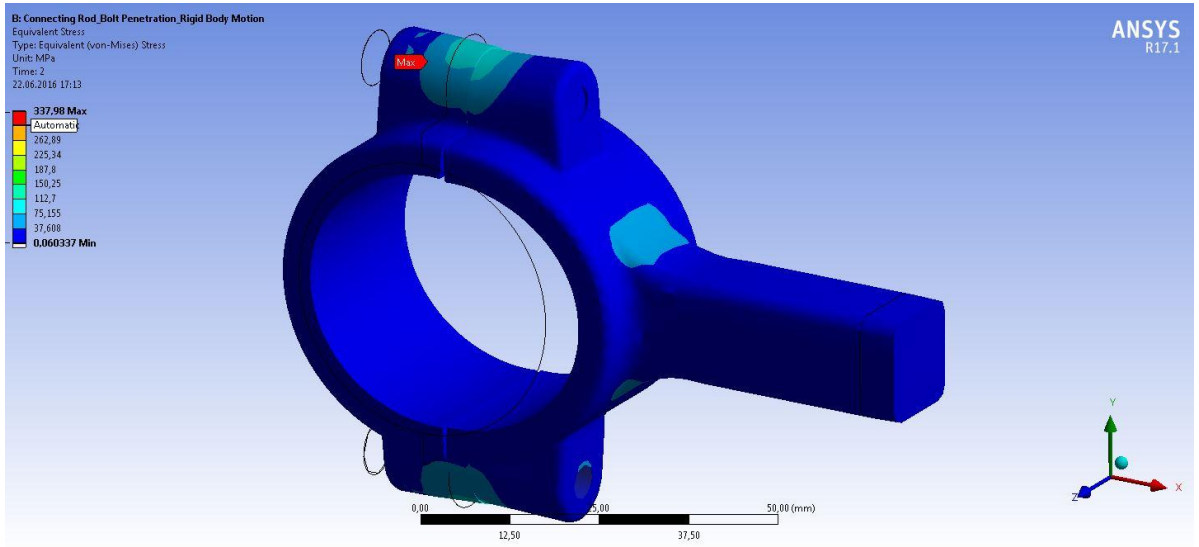
- Day 1
  - 2.) Illustrative Introduction
  - 3.) Connection Groups
  - 4.) Contacts between Surface Bodies
  - 5.) Analysis Settings
  - 6.) Input and Output Files
  - 7.) Contact vs. Target
  - 8.) Force Control vs. Displacement Control
  - 9.) Evaluation of Results
  - 11.) Trim Contact
- Day 2
  - 10.) Contact Elements
  - 12.) Types of Contact
  - 13.) Detection Method
  - 14.) Pinball-Region
  - 15.) Contact Algorithm
  - 16.) Contact Stiffness
  - 17.) Penetrations
- Day 3
  - 18.) Bending Example
  - 19.) Rigid Body Motions
  - 20.) Bonded Contact
  - 21.) Contact Treatment
  - 22.) Convergence Treatment



Ansys contact analysis







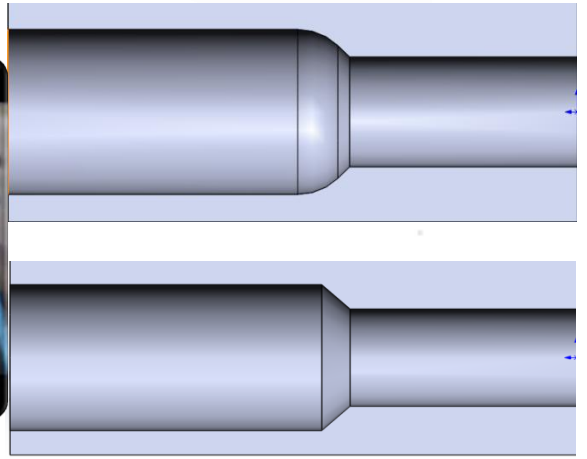
The study of contact analysis were useful as there are a lot of contact surfaces in my Master thesis.

СИБИРСКИЙ ФЕДЕРАЛЬНЫЙ УНИВЕРСИТЕТ
SIBERIAN FEDERAL UNIVERSITY

The Results of the internship

The acquired knowledge proved useful for work in software complex ANSYS. In the course of the internship the following skills have been obtained.

1. The combination of different types of analysis.
2. The creation and configuration of the properties of the material.
3. The work on the setting up of the analysis (Application forces, setting the boundary conditions).
4. The configuration of mesh and mesh setting
5. The Analysis of the results
6. The work in the new SpaceClaim module designed for the topological optimization (Unfortunately this kind of optimization is not suitable for my dissertation)
7. Programming of ACT console integrated into ANSYS and APDL.
8. Solution of contact problems with different types of contacts.



СИБИРСКИЙ ФЕДЕРАЛЬНЫЙ УНИВЕРСИТЕТ
SIBERIAN FEDERAL UNIVERSITY

Thank you for your attention!