SPF SIMULATIONS

Performed at Aircraft Manufacturing Technologies Research and Virtual Simulation Laboratory
by requests of Patrick Romilly and Guillaume Sana (ACB)

Irkutsk State Technical University

Brief task description

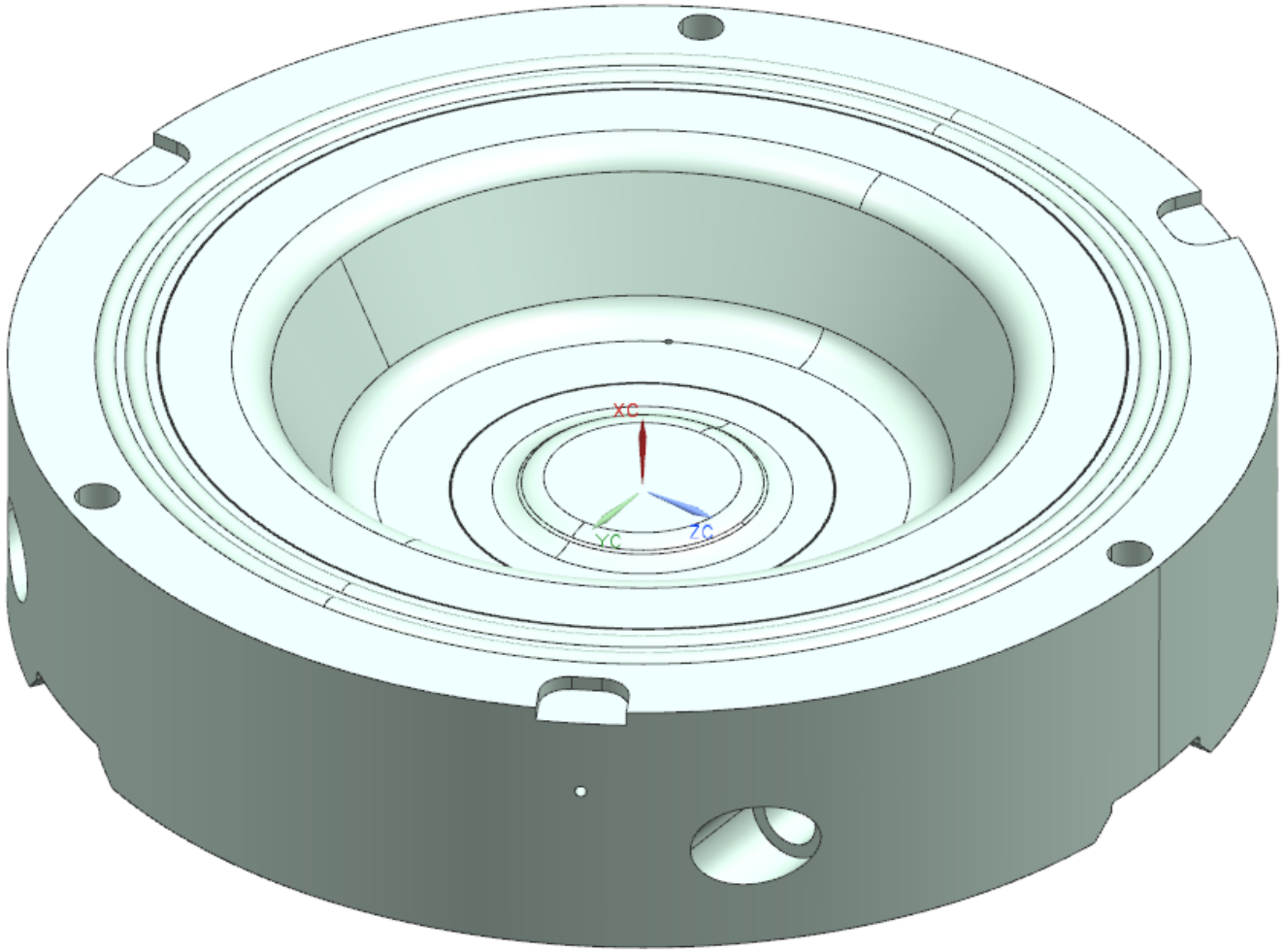
- Initial data provided:
  - tool geometry
  - initial blank thickness
  - material superplastic behavior properties (2 cases with different strain rates, strain rate sensitivities and according flow stresses)
  - friction property (friction coefficient)

- Requested information:
  - forming pressure law (pressure-time dependence)
  - fields of thickness reduction, stresses and strains
  - finite element model and simulation details

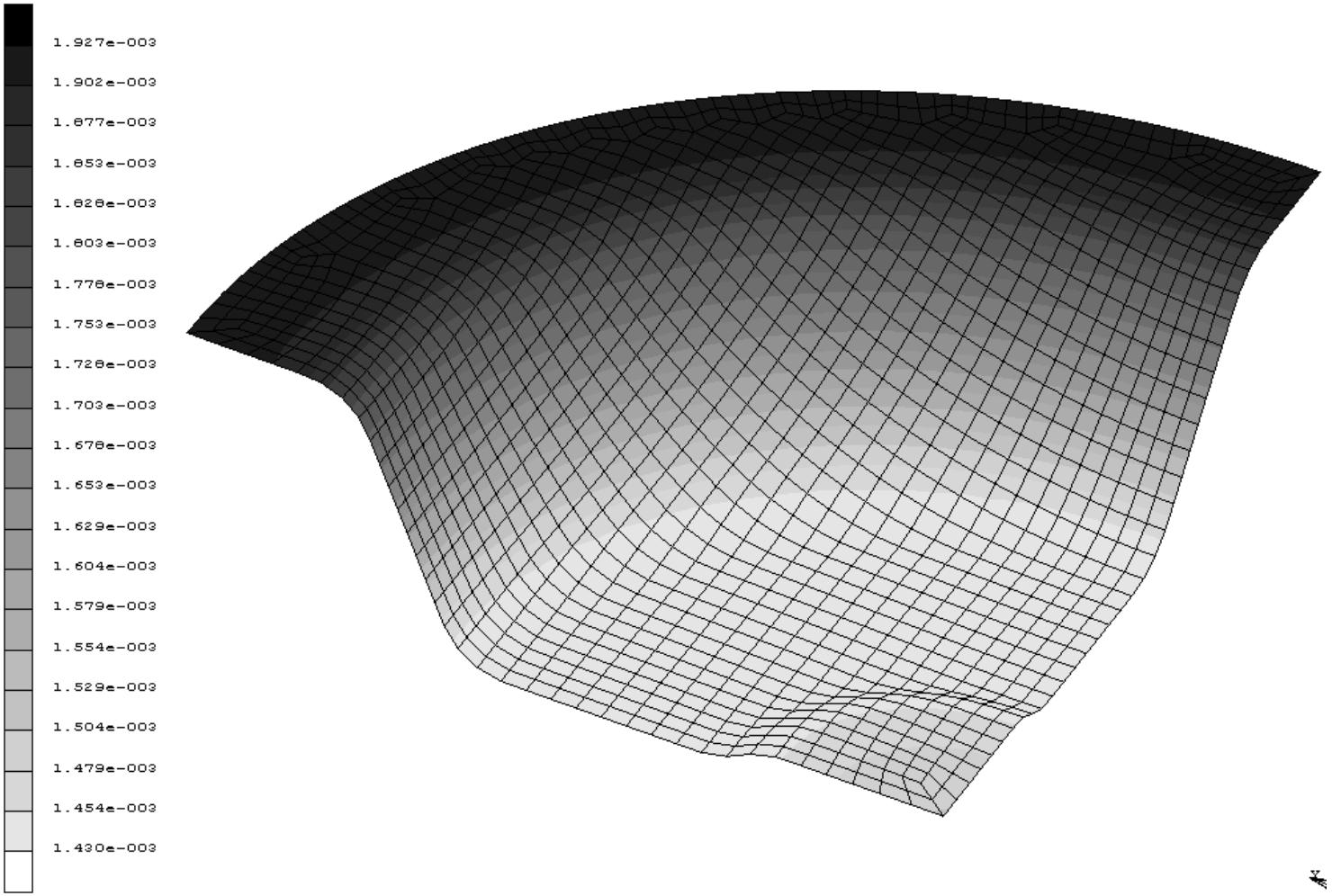
Brief task description (continued) Simulation cases

- Case 1:
  - strain rate sensitivity coefficient 0.5
  - maximum strain rate set to $4 \cdot 10^{-4}\ s^{-1}$
  - superplastic flow stress at strain rate $3 \cdot 10^{-4}\ s^{-1}$ is 8MPa
- Case 2:
  - strain rate sensitivity coefficient 0.37
  - maximum strain rate is $3{,}3 \cdot 10^{-4}\ s^{-1}$
  - superplastic flow equation coefficient $A=400 \cdot 10^{6}$
  - additional check of minimum thickness at the end of forming is requested, minimum thickness is expected to be 1.52 mm

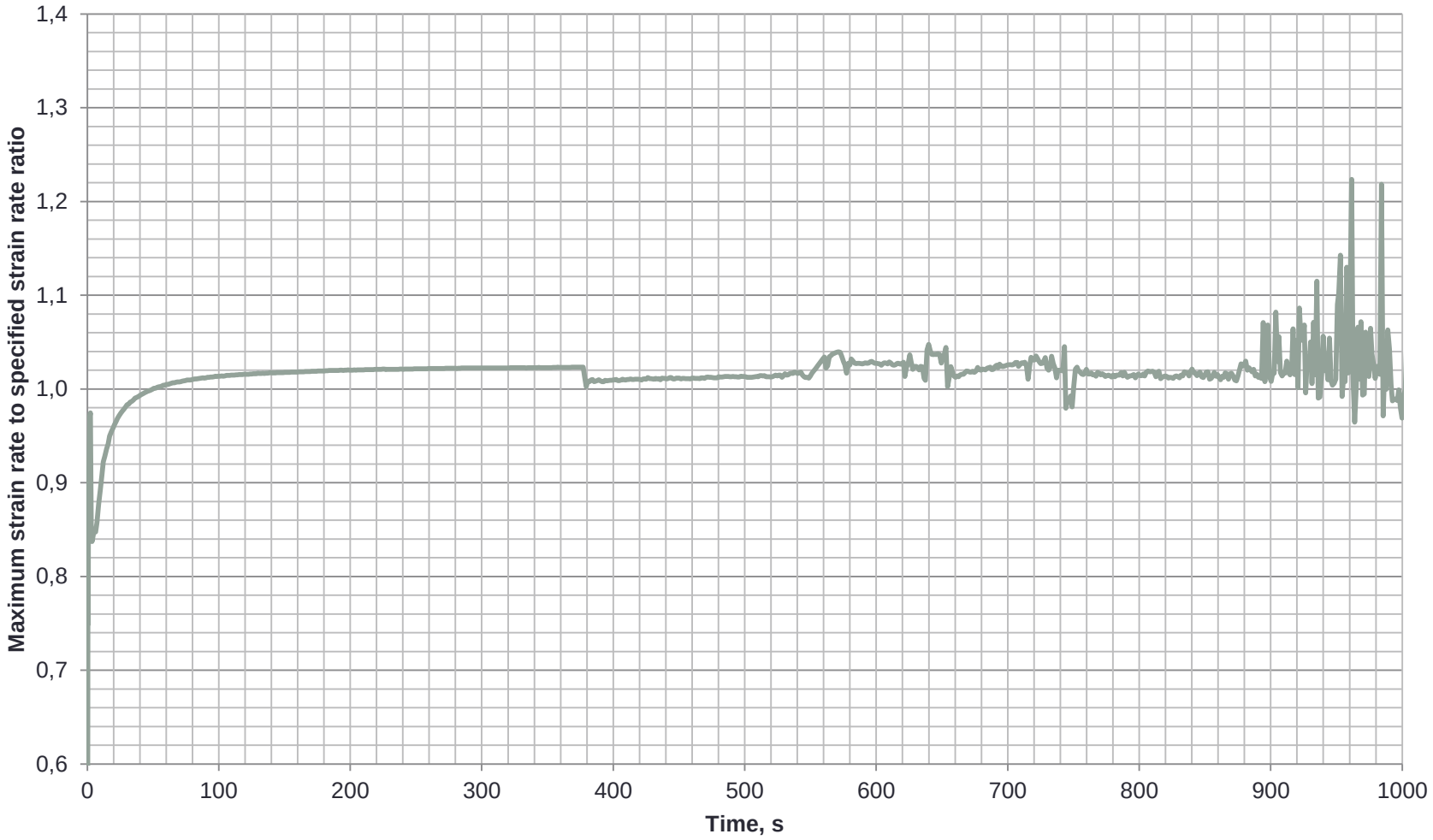
Tool geometry



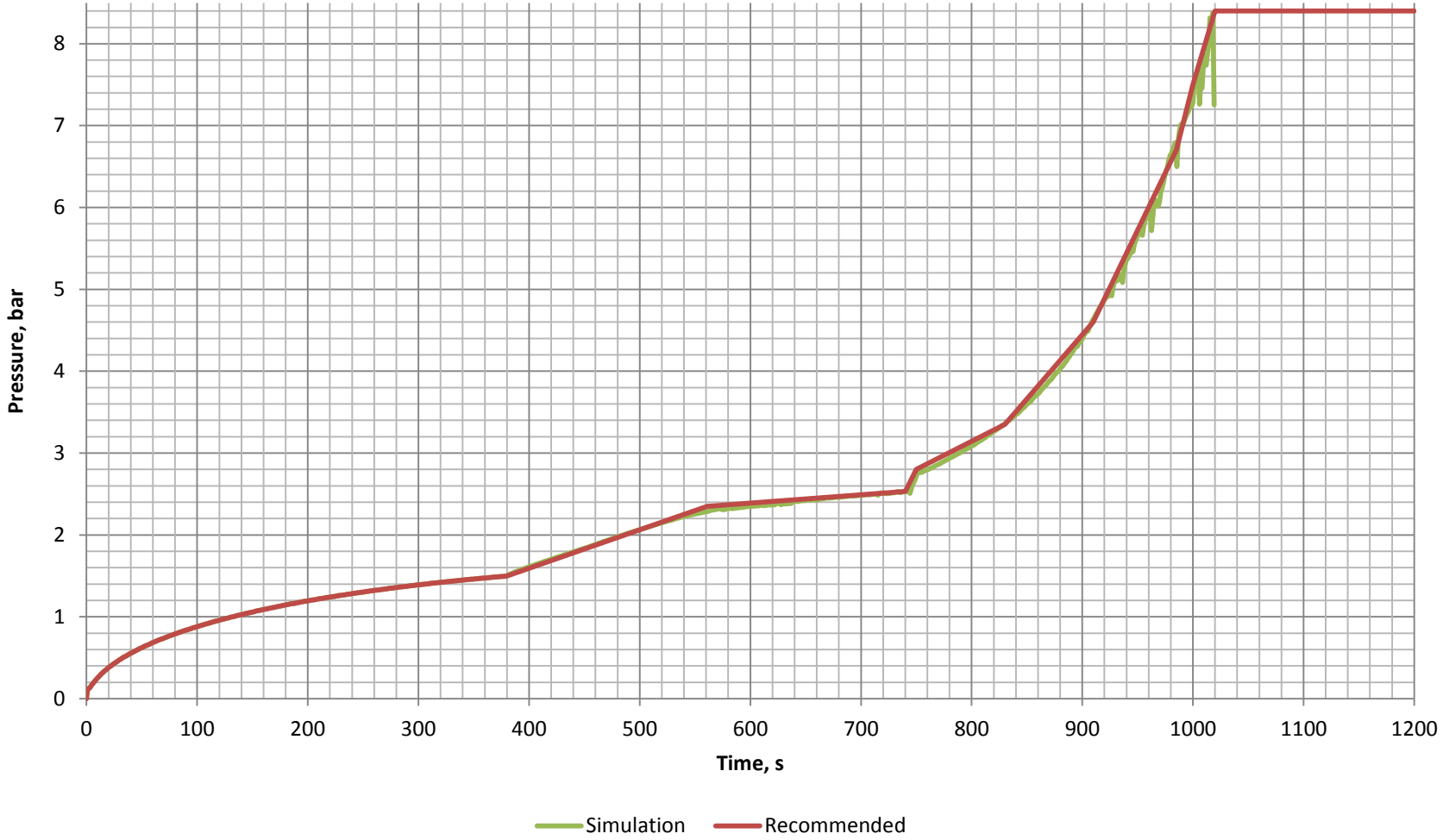
Simulation results, case 1
Field of thickness at the end of simulation



Simulation results, case 1
Maximum strain rate to desired strain rate ratio during simulation



Simulation results, case 1
Pressure-time dependence curve

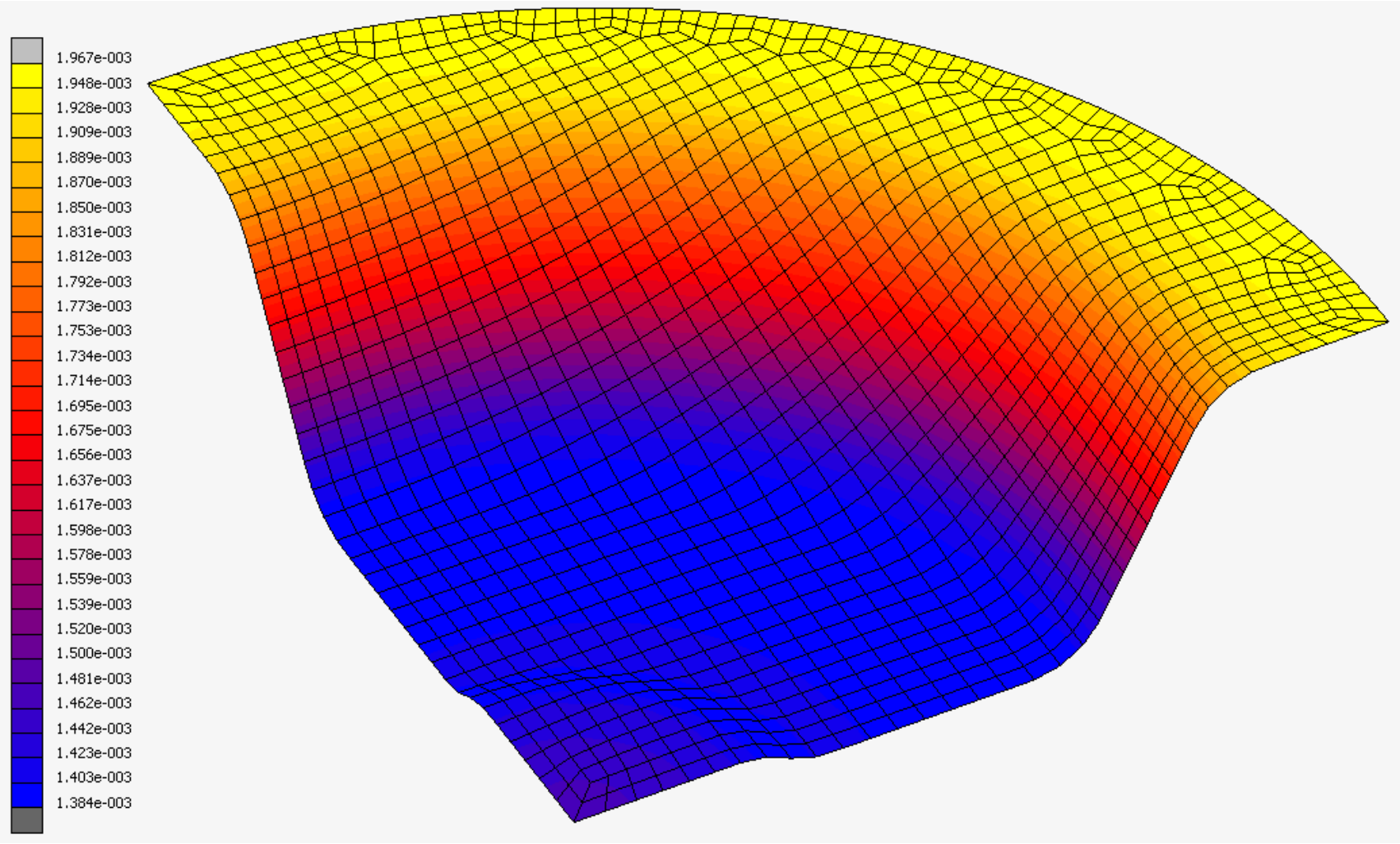


Recommended pressure-time dependence curve is also given, smoothed and supplemented with a constant pressure final calibration stage

Simulation results, case 1 Summary

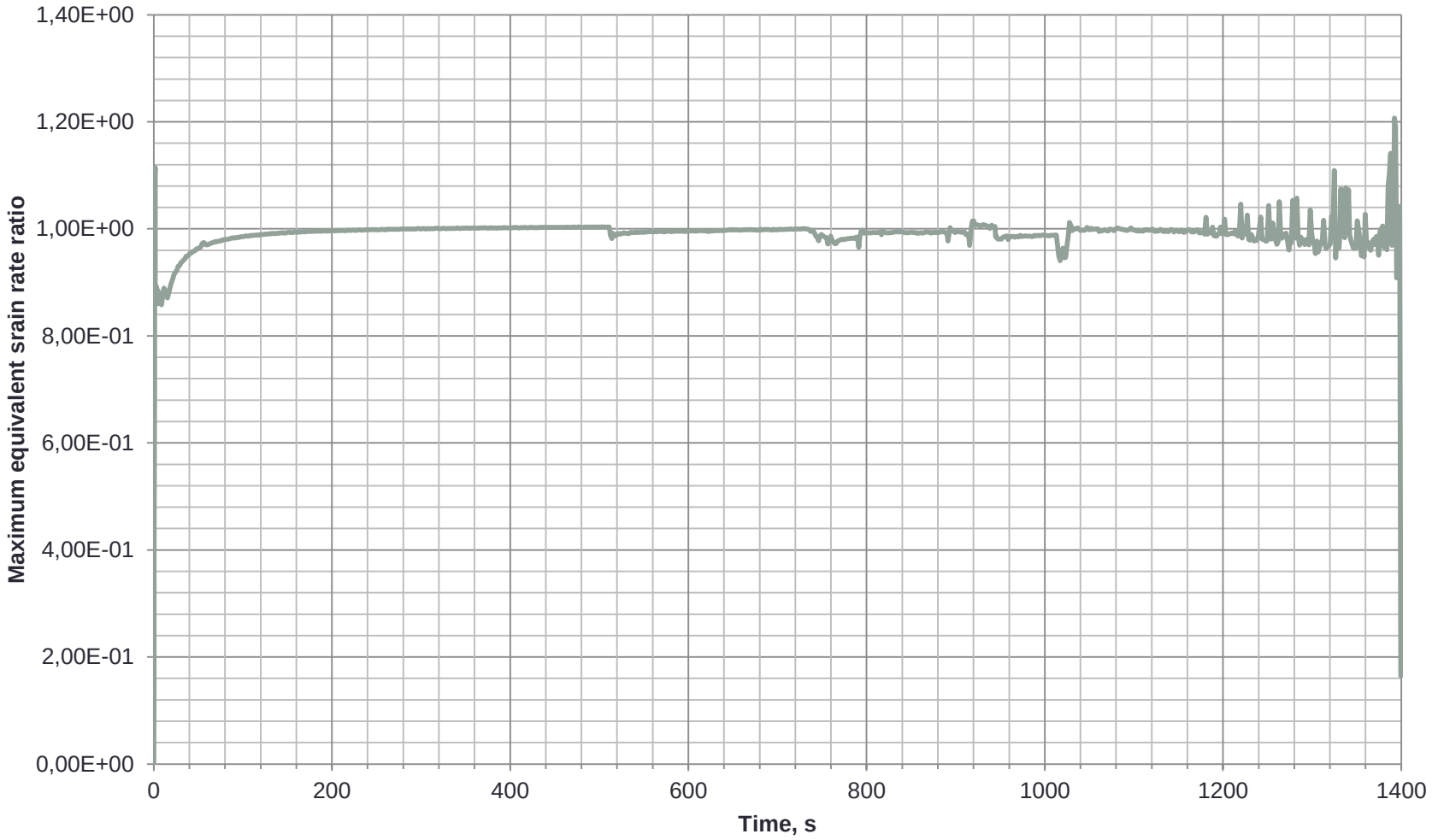
- Forming time (not including pressure unload stage): 1020 seconds
- Maximum required pressure: 8.4 bar
- Maximum plastic strain: 36.1%
- Minimum thickness at the end of simulation: 1.43 mm

Simulation results, case 2
Field of thickness at the end of simulation

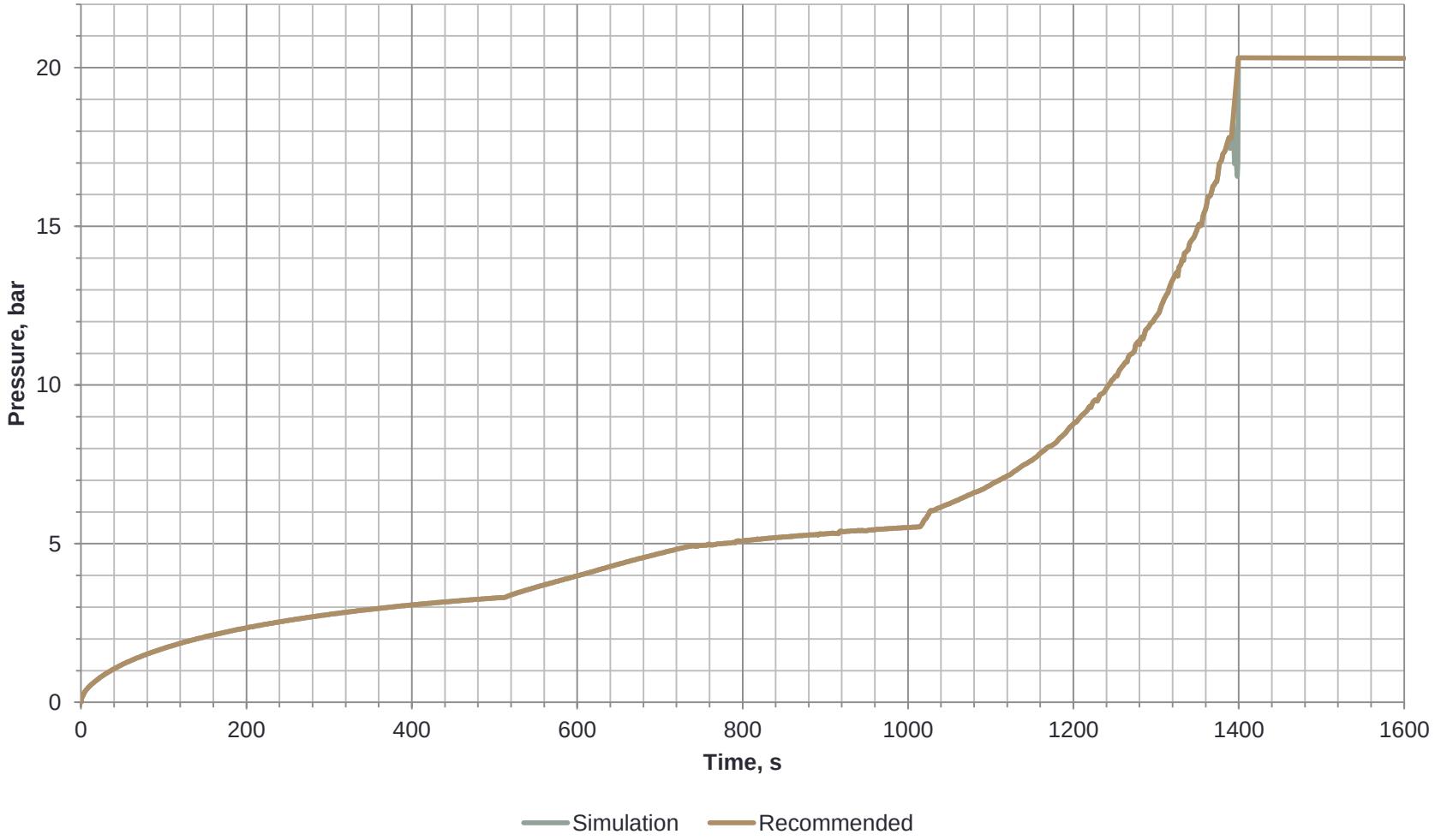


Simulation results, case 2

Maximum strain rate to desired strain rate ratio during simulation



Simulation results, case 2
Pressure-time dependence curve



Recommended pressure-time dependence curve is also given, smoothed and supplemented with a constant pressure final calibration stage

Simulation results, case 2 Summary

- Forming time (not including calibration and pressure unload stages): 1400 seconds
- Maximum required pressure: 20.3 bar
- Maximum plastic strain: 39.4%
- Minimum thickness at the end of simulation: 1.38 mm

- Minimum thickness is different from specified (1.52 mm), additional check was performed (see next)

Simulations results
Additional minimum thickness check

- Simulation case 1 minimum thickness: 1.43 mm
- Simulation case 1 minimum thickness: 1.38 mm

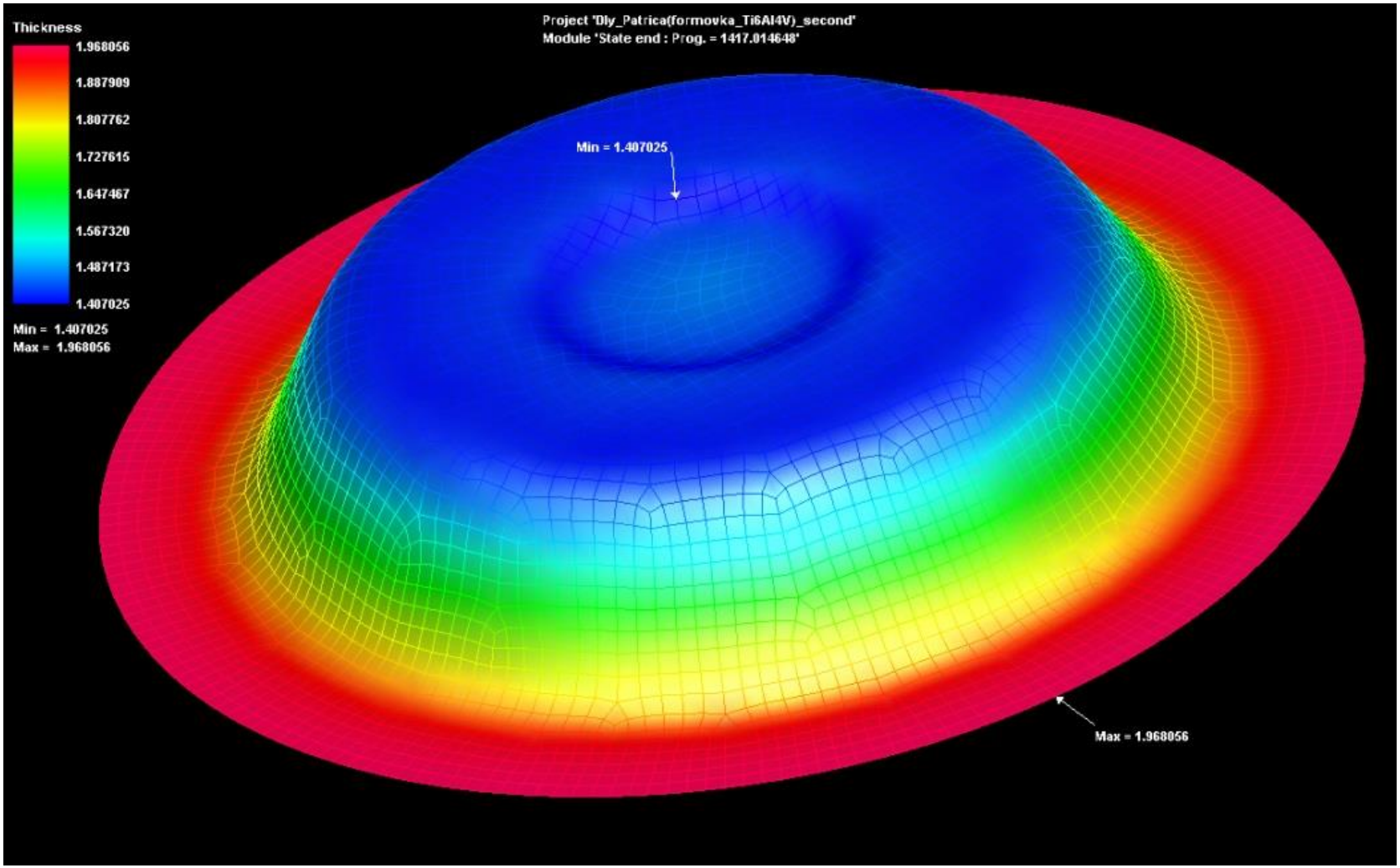
- Expected minimum thickness: 1.52 mm

- Decision: perform control simulation using different CAE software

Control simulation case

- Same initial data as in simulation case 2
- Different CAE software – ESI PAM-STAMP instead of previously used MSC.Marc
- Different FE model:
  - different finite element density
  - different contact formulation
  - different model simplification (more complex model used)

Simulation results, control case
Field of thickness at the end of simulation



Simulation results, control case Summary

- Forming time (not including calibration and pressure unload stages): 1417 seconds
- Maximum plastic strain: 37.8%
- Minimum thickness at the end of simulation: 1.41 mm

- Minimum thickness showed by control simulation is very close to results of cases 1 and 2. Simulation cases may be precise enough.

Summary and conclusion

Case	Strain rate	Forming time	Max. plastic strain	Min. thickness
Case 1	$4 \cdot 10^{-4}\ s^{-1}$	1020	36.1%	1.43 mm
Case 2	$3.3 \cdot 10^{-4}\ s^{-1}$	1400	39.4%	1.38 mm
Control case	$3.3 \cdot 10^{-4}\ s^{-1}$	1417	37.8%	1.41 mm

Minimum thickness showed by control simulation is very close to results of cases 1 and 2. Simulation cases are believed to be of good precision.

If necessary, additional FE-study can be performed using more complex FE-models with finite elements, taking bending strains and stresses into account. This approach can possibly give results of better precision.

Contacts

Aircraft Manufacturing Technologies Research and Virtual Simulation Laboratory

Irkutsk State Technical University
Irkutsk, Russian Federation (local time zone GMT+9)

- Manager: Sergey Osipov
  - tel./fax. +7 (3952) 405540
  - e-mail osipov_sa@istu.edu
- FE-simulation engineer (simulation cases 1 and 2): Ilya Kolmogortsev
  - tel./fax. +7 (3952) 405904
  - e-mail ivk@istu.edu
- FE-simulation engineer (control simulation case): Vladimir Mironenko
  - tel./fax. +7 (3952) 405904
  - e-mail mironenko_vv@istu.edu

Aircraft Manufacturing Technologies Research and Virtual Simulation Laboratory

Laboratory also performs studies in:

- Metal forming processes (sheet metal, stamping, hot forming, SPF, SPF/DB and others):
  - process simulations and studies
  - process design and optimization
  - tools and equipment design and optimization
- Structural engineering and design
  - Strength studies
  - Fatigue studies