Smart consideration of actual ladle status monitored by novel sensors for secondary metallurgy process parameters and ladle maintenance strategies – SmartLadle

Deliverable D3.1 – Set-up and validation of FEM model

Deliverable: **Public** Due 06 / 2022 Lead beneficiary: BFI

Project: Grant Agreement Number:101034017 Duration: 01.07.2021 – 30.06.2024

Birgit Palm, Bernd Kleimt; VDEh-Betriebsforschungsinstitut GmbH (BFI); Düsseldorf, Germany

BFI Excellence in Applied Research | VDEh-Betriebsforschungsinstitut GmbH

Stahl

Content

	Page
› Project summary	3
› Deliverable introduction	4
› Introduction FEM	5
› Geometry setup	6
› Mesh generation	7
› Differential equations	8
› Domain conditions, boundary conditions	14
› Results	15
› Validation	23
› Summary and next steps	24

Project summary

What is the effect of the actual ladle status -new to worn- on steel bath properties? How do e.g. temperature or fluid flow vary with ladle conditions? When is the optimal moment for re-lining?

SmartLadle will provide a solution for online monitoring and dynamic incorporation of actual ladle status for process control. A soft sensor for ladle status shall be developed, supported by a smart sensor for detecting refractory wear and thermal status. Measurement data, models and advisory tools shall provide information for decision making to operators to adapt ladle metallurgy process parameters to actual ladle status and decide about maintenance actions.

Deliverable introduction

Deliverable 3.1 describes work performed in Task 3.1 and provides information about the set-up of a FEM model to calculate temperature and thermomechanical stresses in ladle refractory and steel shell, and the validation of this model using measured temperature values from SWG ladle.

For using the measured temperature values (of the smart sensor developed within SmartLadle) to give an assessment about ladle wear and thermal status, a systematic investigation of temperature evolution is needed. Therefore, numerical modelling work using Finite Element Method (FEM) is applied. An existing model is enhanced to consider wear influences by e.g. implementing changing boundary conditions for reproducing the decrease of working lining thickness.

To ensure a best fitting numerical model, temperature measurements inside the ladle refractory at different positions will be used to adapt the model. Such ladle instrumentation has been successfully realised by the partners involved in previous projects, but has faced some difficulties and is not yet completed within SmartLadle.

Introduction FEM

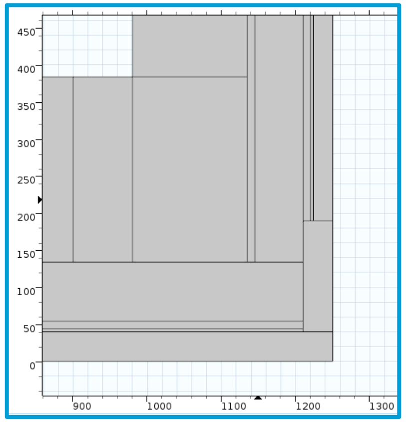
VDEh-Betriebsforschungsinstitut GmbH

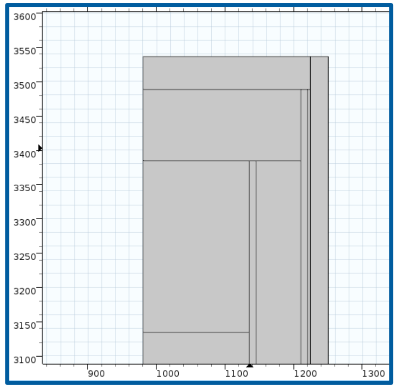
The Finite Element Method is a numerical technique for the solution of differential equations. At first the continuous calculation domain is divided into a set of discrete (finite) subdomains, usually called elements (mesh discretization). Each subdomain is represented by a set of element equations to the original problem, followed by systematically recombining all sets of element equations into a global system of equations for the final calculation. A common approach for performing numerical analysis includes the following steps:

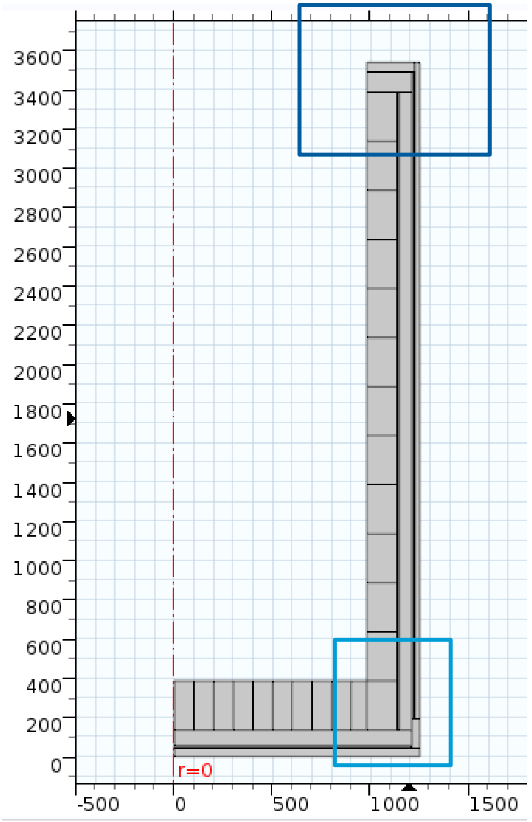
- geometry setup within the software program coded with FEM algorithm
- mesh generation for dividing a complex problem into small elements
- definition of differential equations to solve the numerical problem
- definition of domain conditions (material properties), initial values and boundary conditions
- solution of differential equations.

Geometry setup

- Geometry setup performed in 2D axial symmetry using technical drawings of the SWG ladle with the simplification of assuming straight ladle wall
- Thickness of layers parametrised to consider wear influence and to enable variation of geometry layout

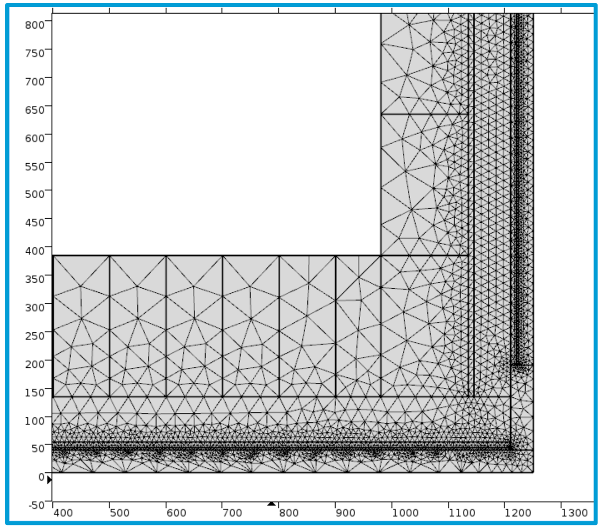


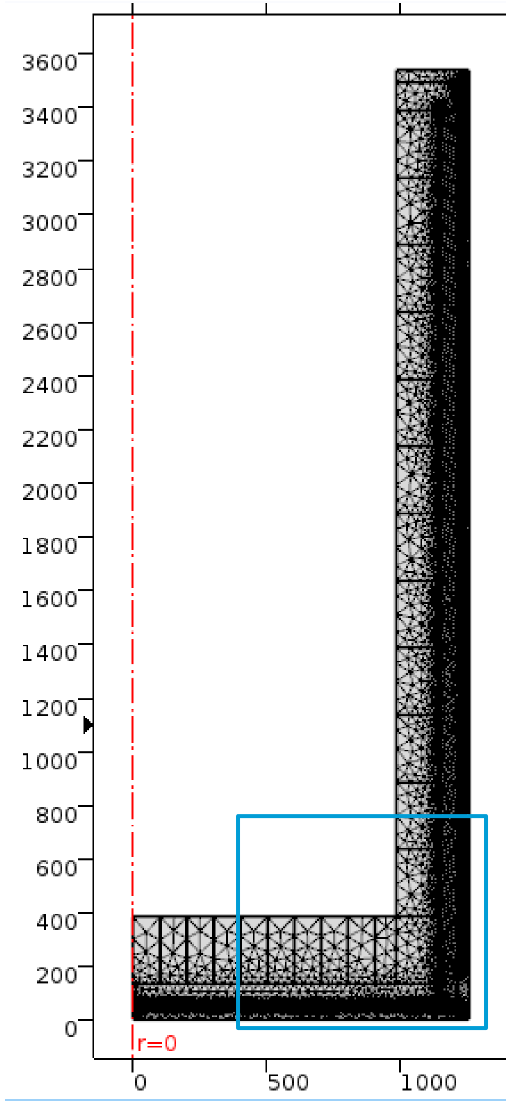




Mesh generation

- Free triangular mesh was created using normal mesh size
- Might be adapted to finer size if necessary





Differential equations

- The heat transfer by thermal conductivity process in the refractory lining is described by the Fourier's differential equation:

$$c_p \cdot \rho \cdot \frac{\partial T}{\partial t} = \nabla \cdot (\lambda \cdot \nabla T) = \frac{\partial}{\partial x}\left(\lambda \cdot \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(\lambda \cdot \frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(\lambda \cdot \frac{\partial T}{\partial z}\right) \qquad (1)$$

  where c_p is the heat capacity, ρ is the density, T is the temperature, t is the time, ∇ is the del operator, λ is the thermal conductivity, and x, y and z are the coordinates

- Temperature differences in the refractory material cause deformations → lead to strains and stresses
- Deformations can be computed as a solution of partial differential equations; the strains and stresses can be computed based on the obtained deformations

Differential equations

VDEh-Betriebsforschungsinstitut GmbH

- The stresses in each point of the refractory lining can be described by a stress tensor σ. In the Cartesian coordinates system σ has the following form:

$$\sigma := \begin{pmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{pmatrix} \qquad (2)$$

  σ_i are the normal stresses and τ_i the shear stresses. The tensor is symmetric.

- The strains are described by the symmetric strain tensor ε:

$$\varepsilon := \begin{pmatrix} \varepsilon_{xx} & \varepsilon_{xy} & \varepsilon_{xz} \\ \varepsilon_{yx} & \varepsilon_{yy} & \varepsilon_{yz} \\ \varepsilon_{zx} & \varepsilon_{zy} & \varepsilon_{zz} \end{pmatrix} \qquad (3)$$

- The Elements ε_{ij} of the strain tensor are computed as followed:

$$\varepsilon_{xx} = \frac{\partial u}{\partial x} \qquad \varepsilon_{yy} = \frac{\partial v}{\partial y} \qquad \varepsilon_{zz} = \frac{\partial w}{\partial z}$$

$$\varepsilon_{xy} = \varepsilon_{yx} = \frac{1}{2}\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right) \qquad \varepsilon_{yz} = \varepsilon_{zy} = \frac{1}{2}\left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}\right) \qquad \varepsilon_{xz} = \varepsilon_{zx} = \frac{1}{2}\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right) \qquad (4)$$

u, v, w being velocity components

Differential equations

- For a Temperature change ΔT in the refractory material the strains are defined as:

$$\varepsilon_{xx} = \varepsilon_{yy} = \varepsilon_{zz} = \alpha \cdot \Delta T \qquad \varepsilon_{xy} = \varepsilon_{yz} = \varepsilon_{xz} = 0 \tag{5}$$

α is the material specific thermal expansion coefficient

- The different parts of the refractory material are constraining each other by changing their volume. This leads to the thermal stresses σ_i and τ_{ik}. The relation of the different factors is described as:

$$\varepsilon_{ik} = \frac{1}{2 \cdot G}\left(\sigma_{ik} - \frac{1}{1-\nu} \cdot s \cdot \delta_{ik}\right) \tag{6}$$

G is the rigidity modulus which is related to Young's modulus E as follows:

$$G = \frac{1}{2} \cdot \frac{E}{1-\nu} \tag{7}$$

ν is the Poisson's ratio which describes the proportion from transverse strain to axial strain. s is the sum of the normal stresses, $s = \sigma_x + \sigma_y + \sigma_z$. δ_{ik} is the displacement vector.

Differential equations

- The stress sum s depends on the volume dilatation e = $\varepsilon_x+\varepsilon_y+\varepsilon_z$:

$$e = \frac{(1-2\nu)\cdot s}{(1+\nu)\cdot 2\cdot G} + 3\cdot\alpha\cdot\Delta T \qquad (8)$$

- Solving equation (6) to σ_{ik} using (8) and (4) results in a system of partial differential equations for deformations where Δ is called the Laplace Operator:

$$\Delta u = \alpha\cdot\frac{2\cdot(1+\nu)}{1-2\nu}\cdot\frac{\partial T}{\partial x} - \frac{1}{1-2\nu}\cdot\frac{\partial e}{\partial x}$$

$$\Delta v = \alpha\cdot\frac{2\cdot(1+\nu)}{1-2\nu}\cdot\frac{\partial T}{\partial y} - \frac{1}{1-2\nu}\cdot\frac{\partial e}{\partial y} \qquad (9)$$

$$\Delta w = \alpha\cdot\frac{2\cdot(1+\nu)}{1-2\nu}\cdot\frac{\partial T}{\partial z} - \frac{1}{1-2\nu}\cdot\frac{\partial e}{\partial z}$$

- This system of partial differential equations is a coupled system as through the volume dilatation e and the strains ε_i (see (4)) a link between the equations exists.
A simultaneous computation of all three equations is now possible.

Differential equations

- The system of linear equations can be written as followed:

$$\begin{pmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_x \\ \tau_y \\ \tau_z \end{pmatrix} = \frac{E}{(1+\nu)\cdot(1-2\nu)} \cdot \begin{pmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} \end{pmatrix} \cdot \begin{pmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ 2\varepsilon_{xy} \\ 2\varepsilon_{yz} \\ 2\varepsilon_{xz} \end{pmatrix} \qquad (10)$$

- For the computation of the thermal stresses, it is thus necessary to compute a coupled system of the three stress differential equations (9) and the Fourier differential equation (1).
  First, every step has to be determined by the Fourier’s differential equation, to achieve the temperature distribution in the refractory material. With the computed temperature field, the deformations can be determined (9) which leads to the strains (4) and stresses (10). Thus, for every step to the computation of thermal stresses, the four equations (1), (4), (9) and (10) have to be solved.

Differential equations

› As a characteristic measure for the stresses, the von Mises stress $\sigma_{v.mises}$ is used:

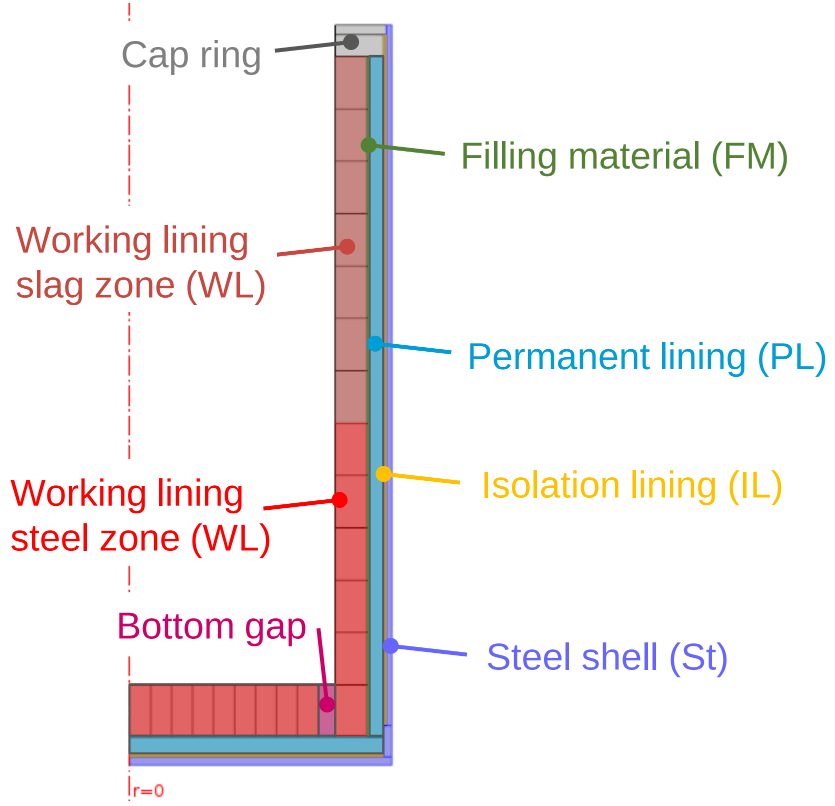
$$\sigma_{v.mises} := \sqrt{\sigma_x^2 + \sigma_y^2 + \sigma_z^2 - \sigma_x\sigma_y - \sigma_y\sigma_z - \sigma_x\sigma_z + 3\cdot\left(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{xz}^2\right)} \qquad (11)$$

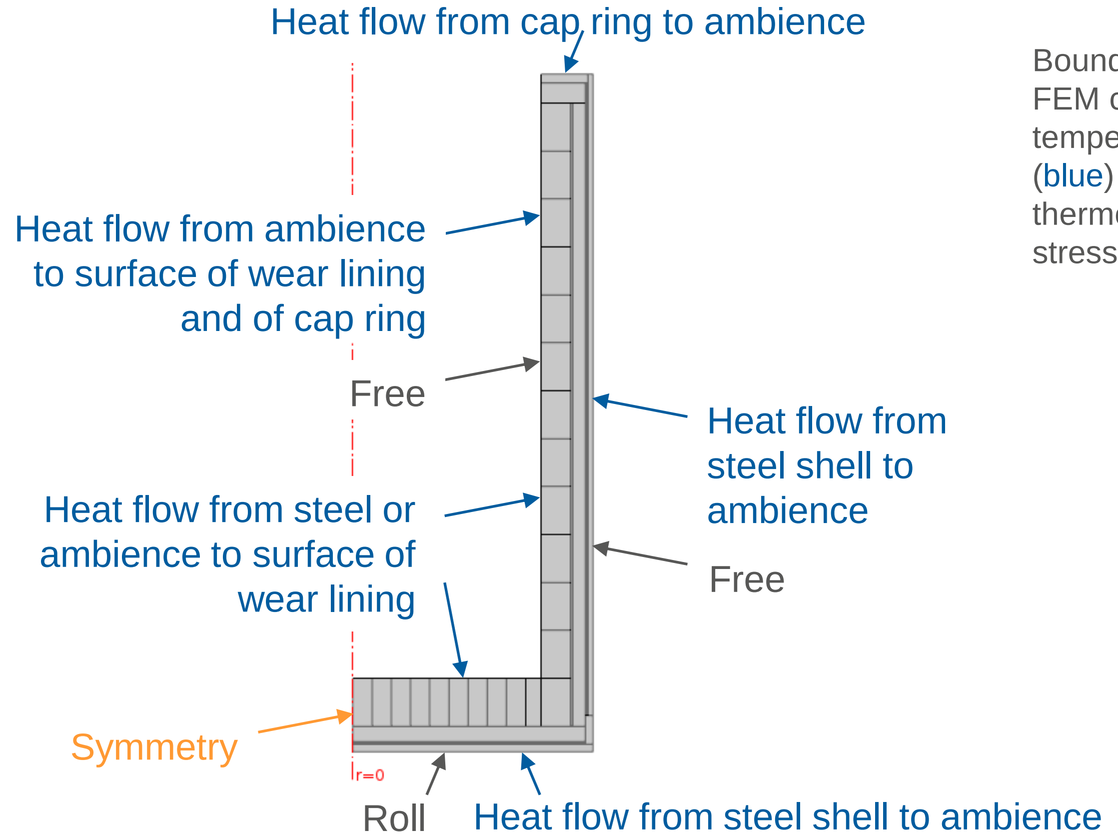
› The numerical solution of the system of coupled differential equations (9) and the Fourier's differential equations (1) is computed with the Finite Element Method (FEM) which is part of the commercial FEM tool COMSOL Multiphysics.

Domain conditions, boundary conditions

VDEh-Betriebsforschungsinstitut GmbH

- Domain conditions (left) have been defined using the results concerning material properties that were gained in Task 1.1
- Boundary conditions (right)) have been defined using the process data obtained in Task 1.1, with two simplifications: bath height remains constant over whole ladle cycle, T_{steel} = 1600°C



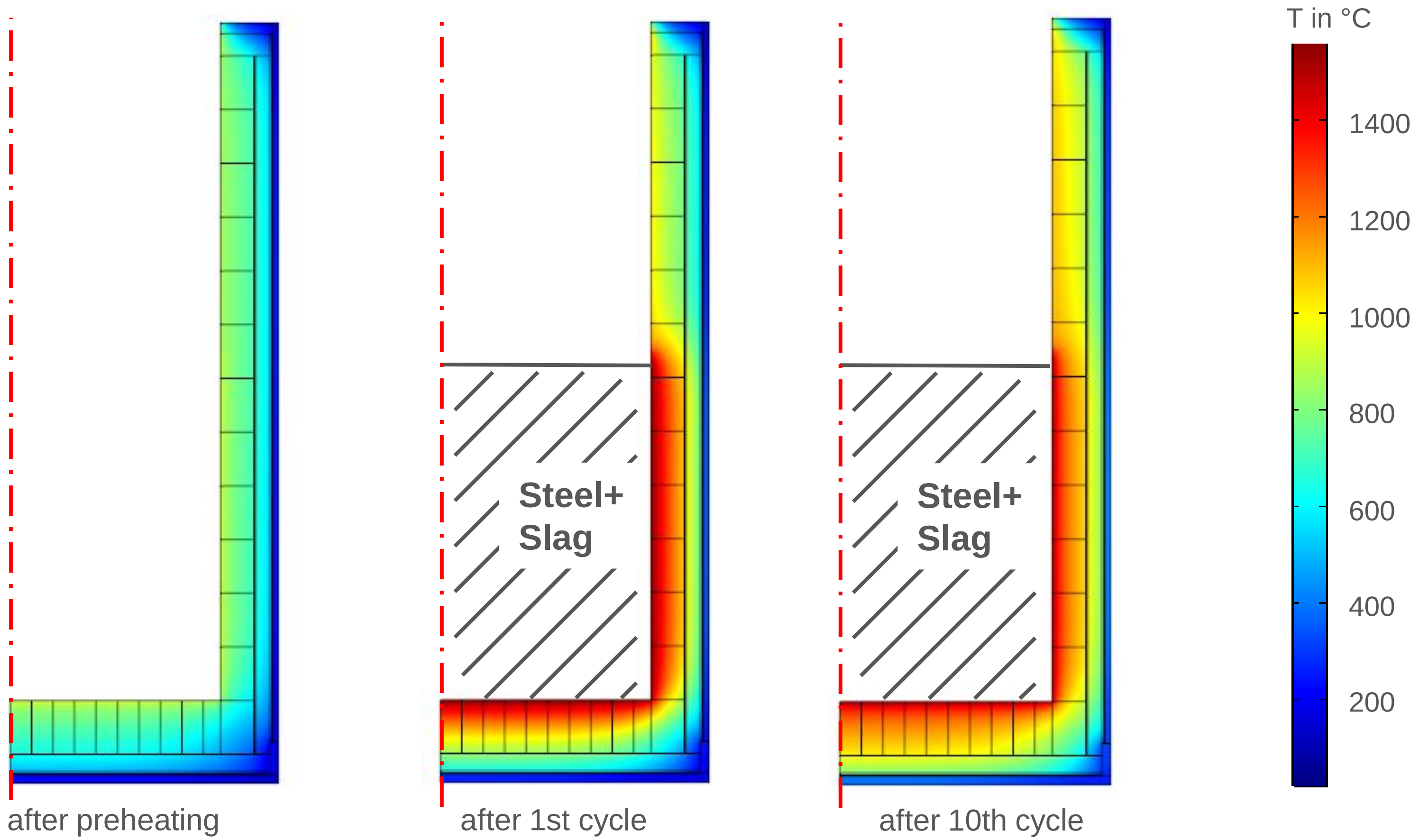


Boundary conditions for FEM calculation of temperature distribution (blue) and thermomechanical stresses (grey)

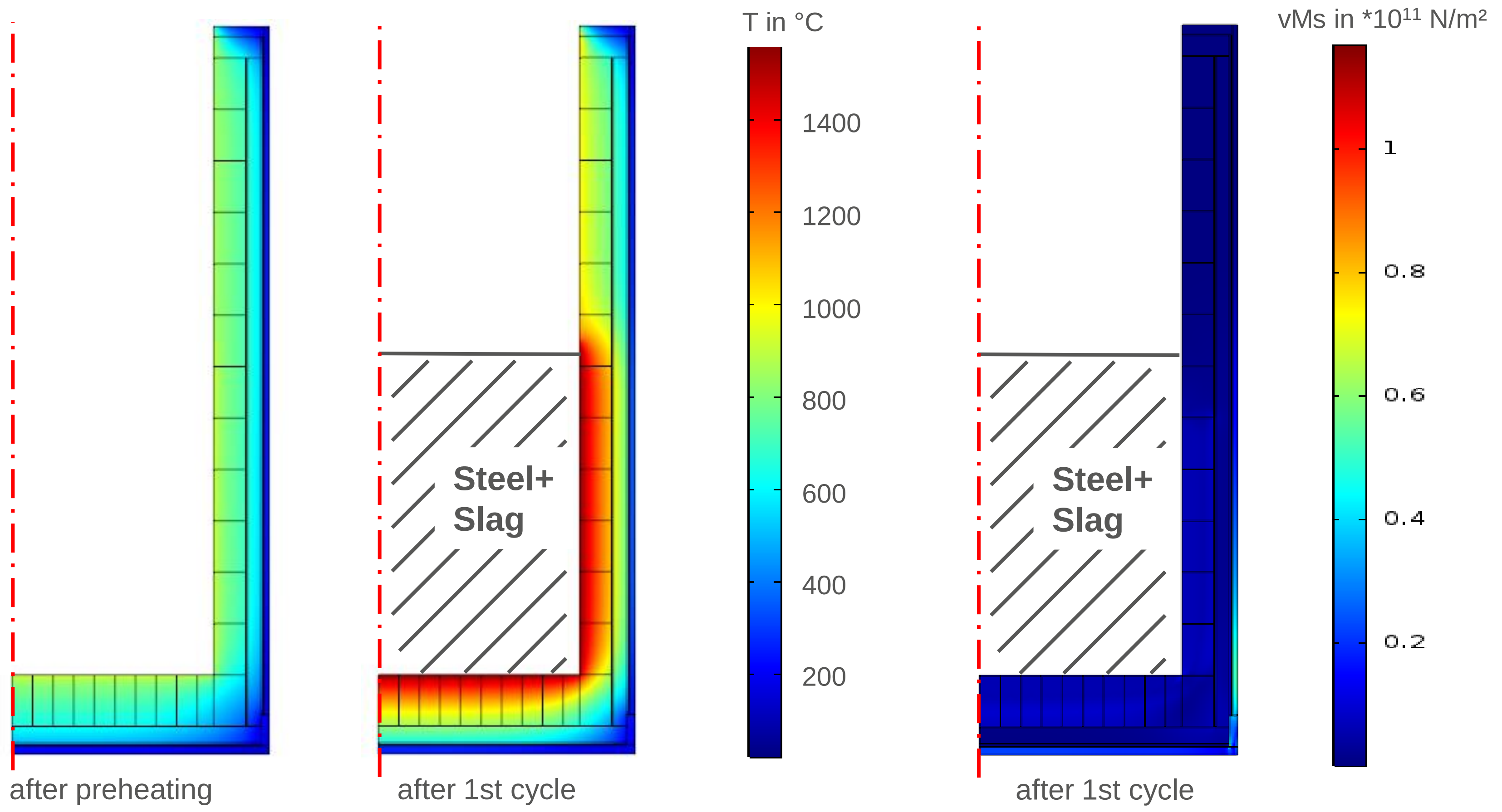
Results – Temperature distribution

VDEh-Betriebsforschungsinstitut GmbH

› Transient solution of differential equations, considering heat transfer in solids, heat flow in and out (see boundary conditions) and heat radiation from surface to surface as well as solid state mechanics



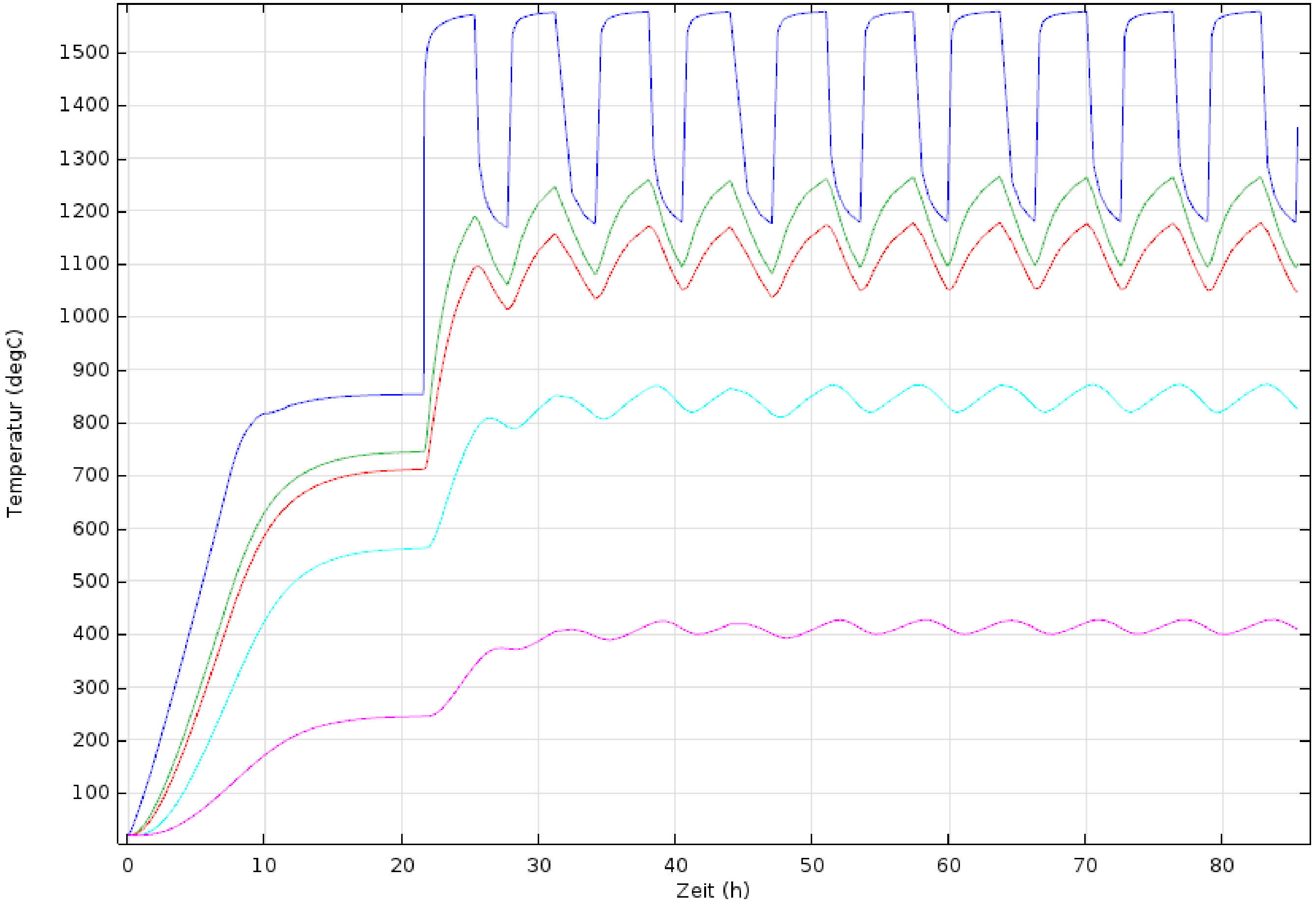
Results – Temperature and von Mises stress distribution

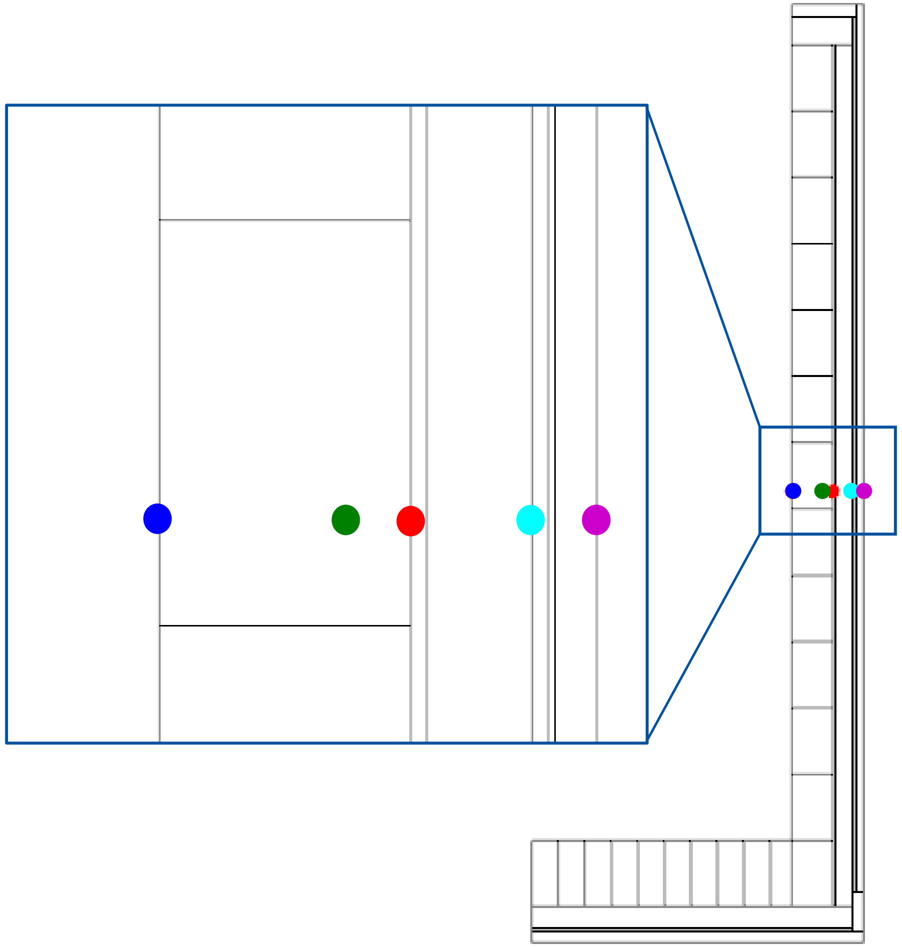


Temperature distribution after preheating and after first cycle (as from slide before) and von Mises stress distribution after first cycle (right)

Results – Temperature over time

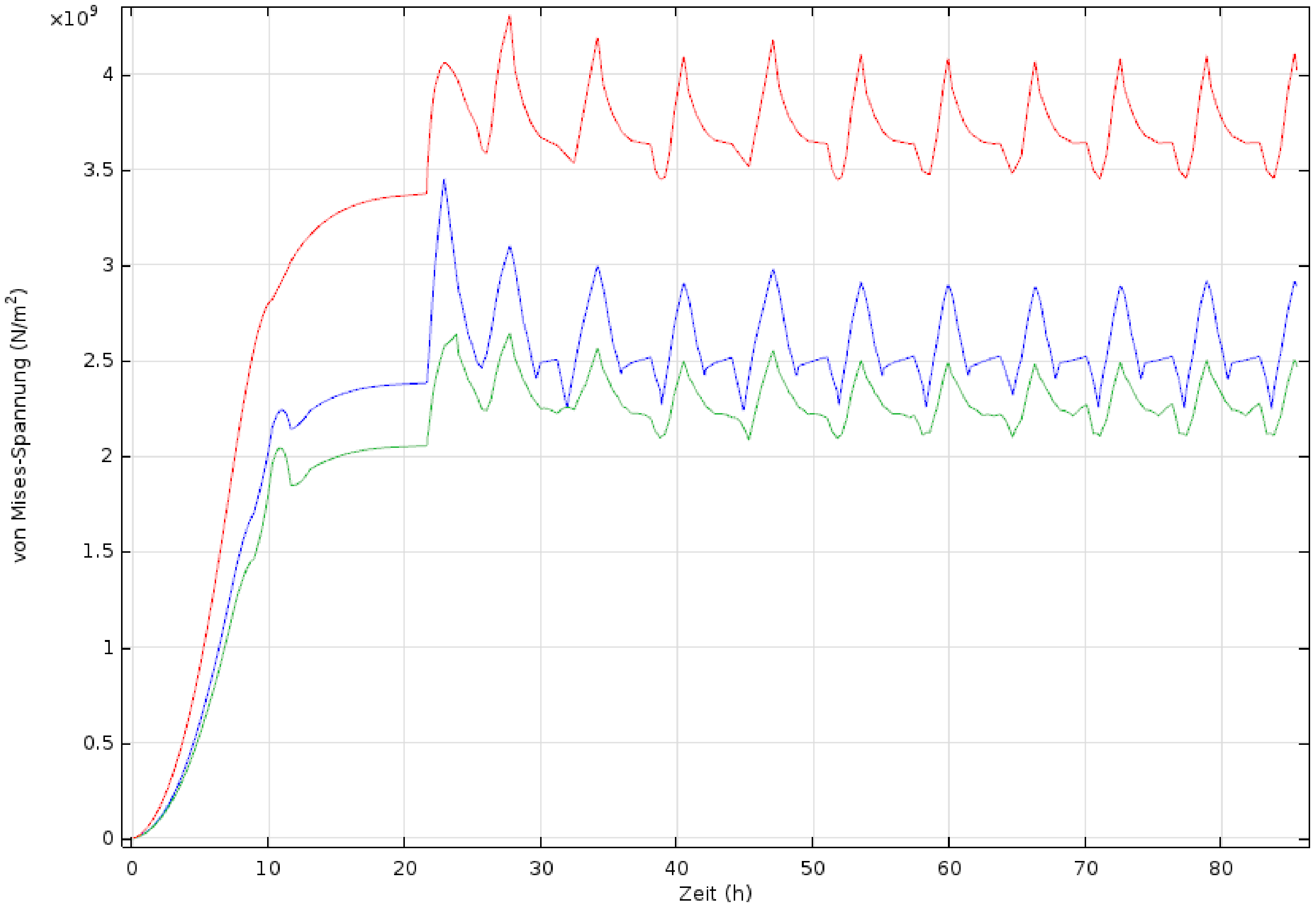
BFI Excellence in Applied Research | VDEh-Betriebsforschungsinstitut GmbH

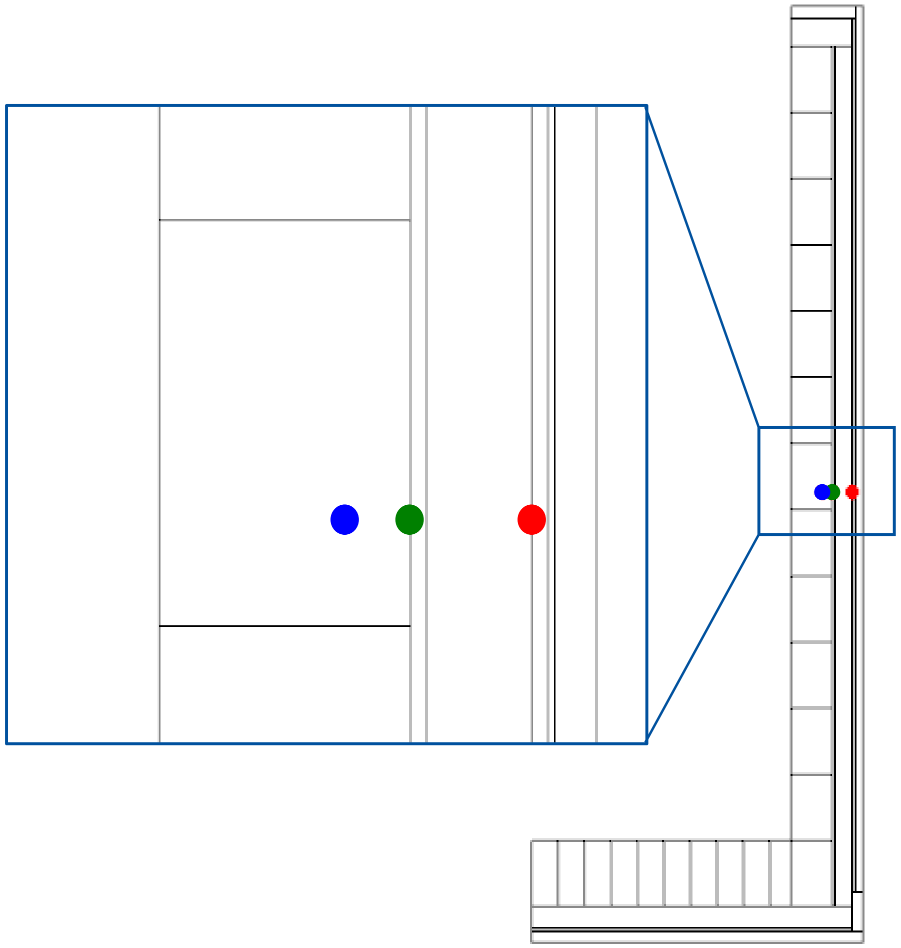




Temperature over time for preheating and first 10 cycles at different positions (see right)

Results – von Mises stress over time

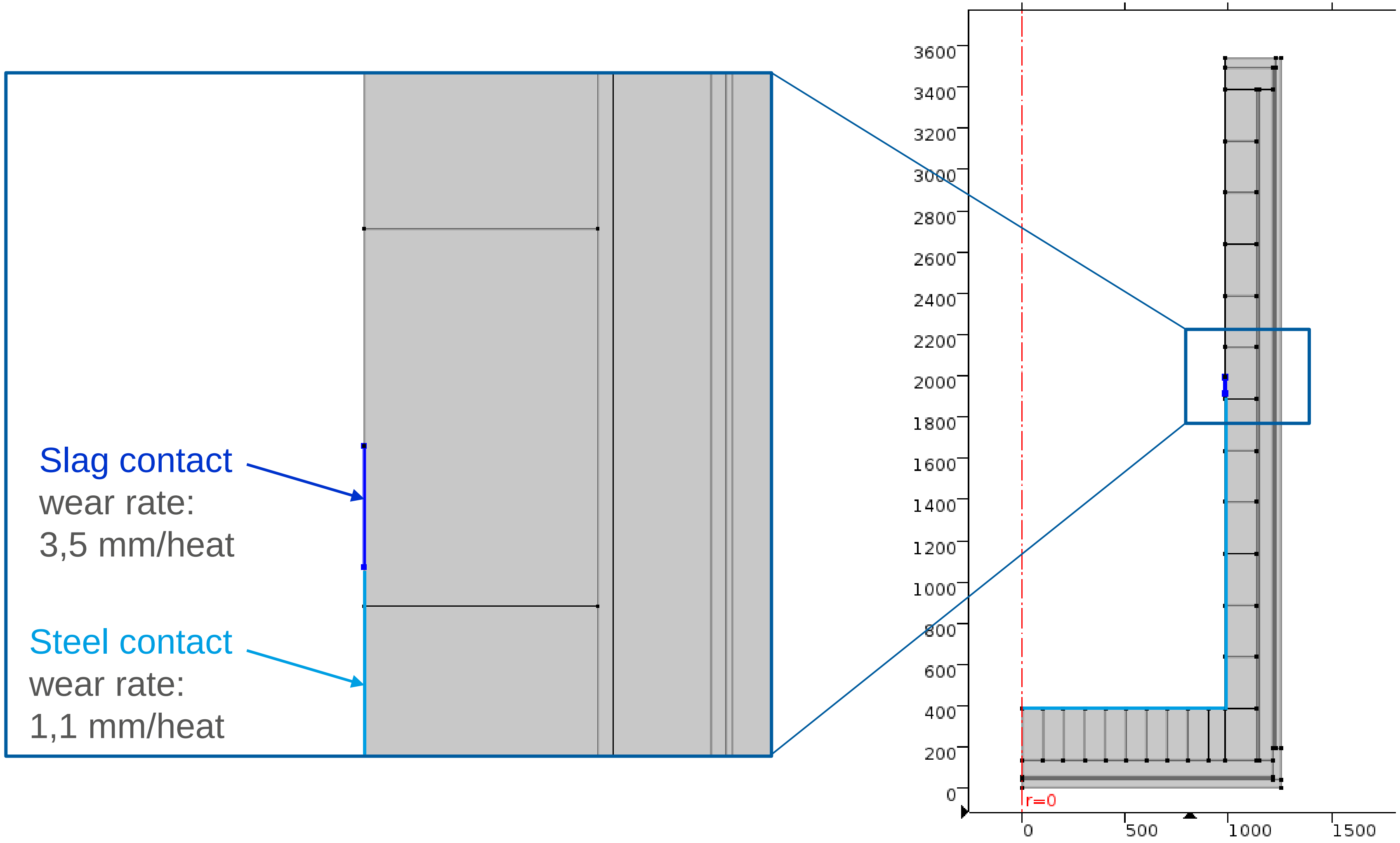




von Mises stress over time for preheating ad first 10 cycles at different positions (see right)

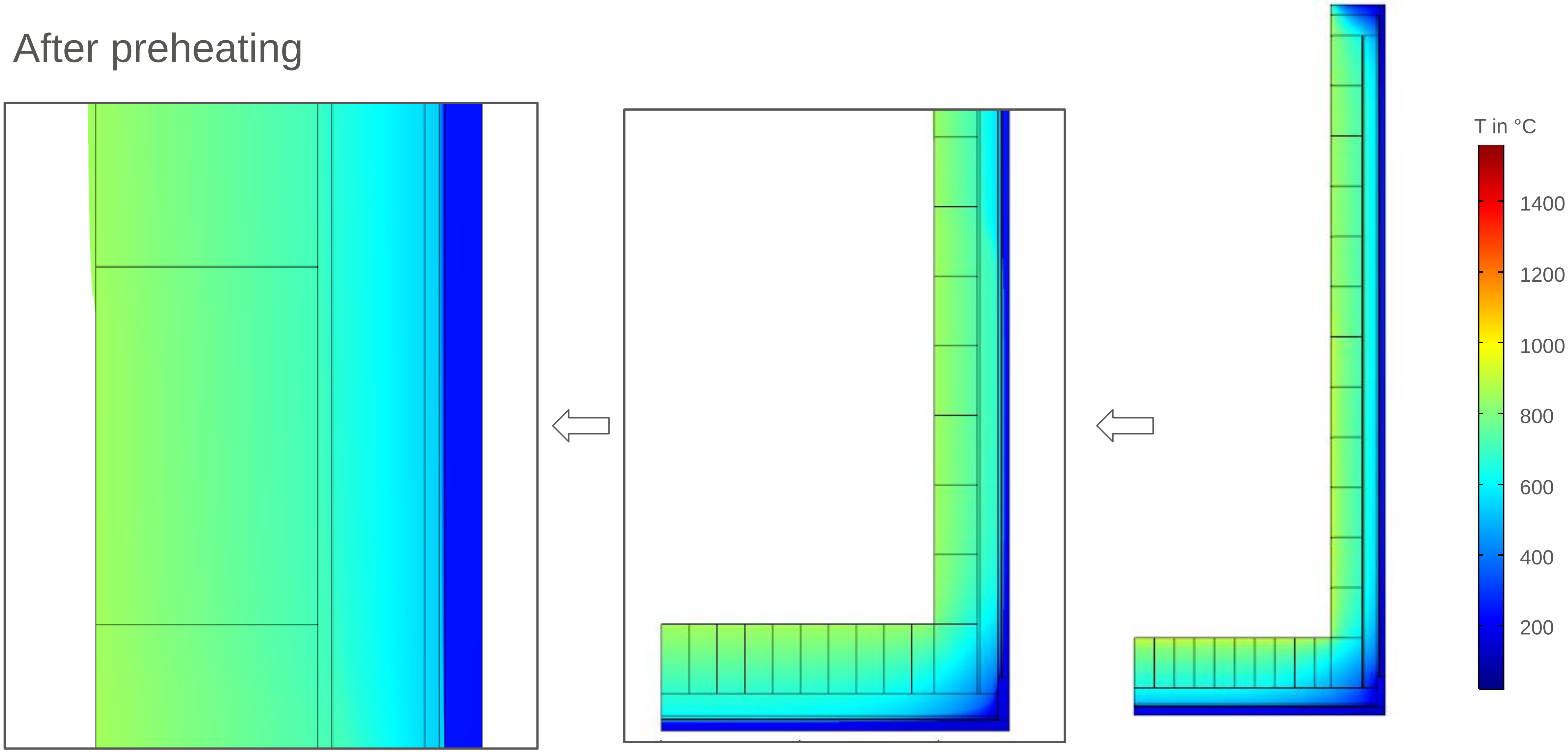
Results – Temperature distribution with wear

- Wear considered by constant displacement during steel cycles



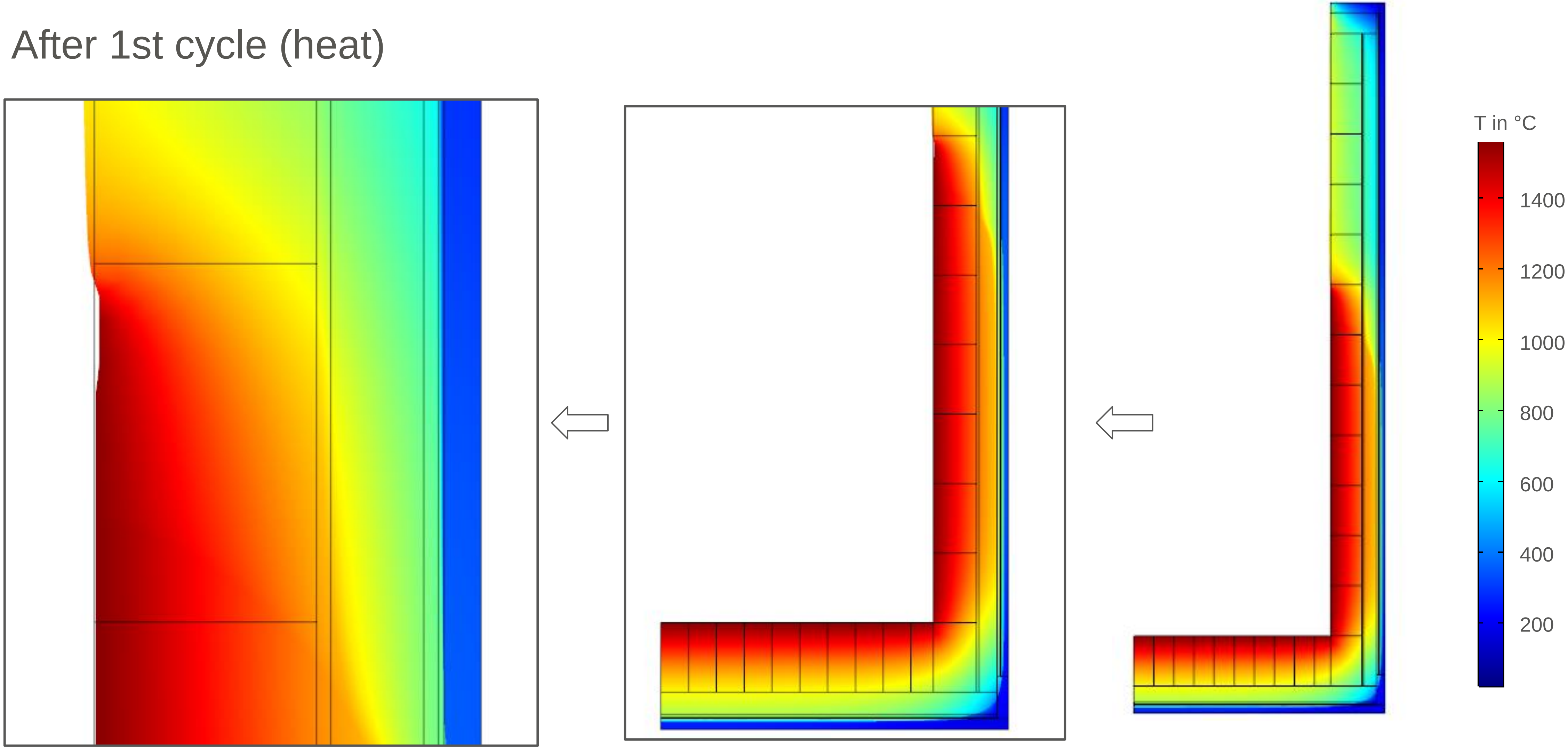
Results – Temperature distribution with wear

- After preheating



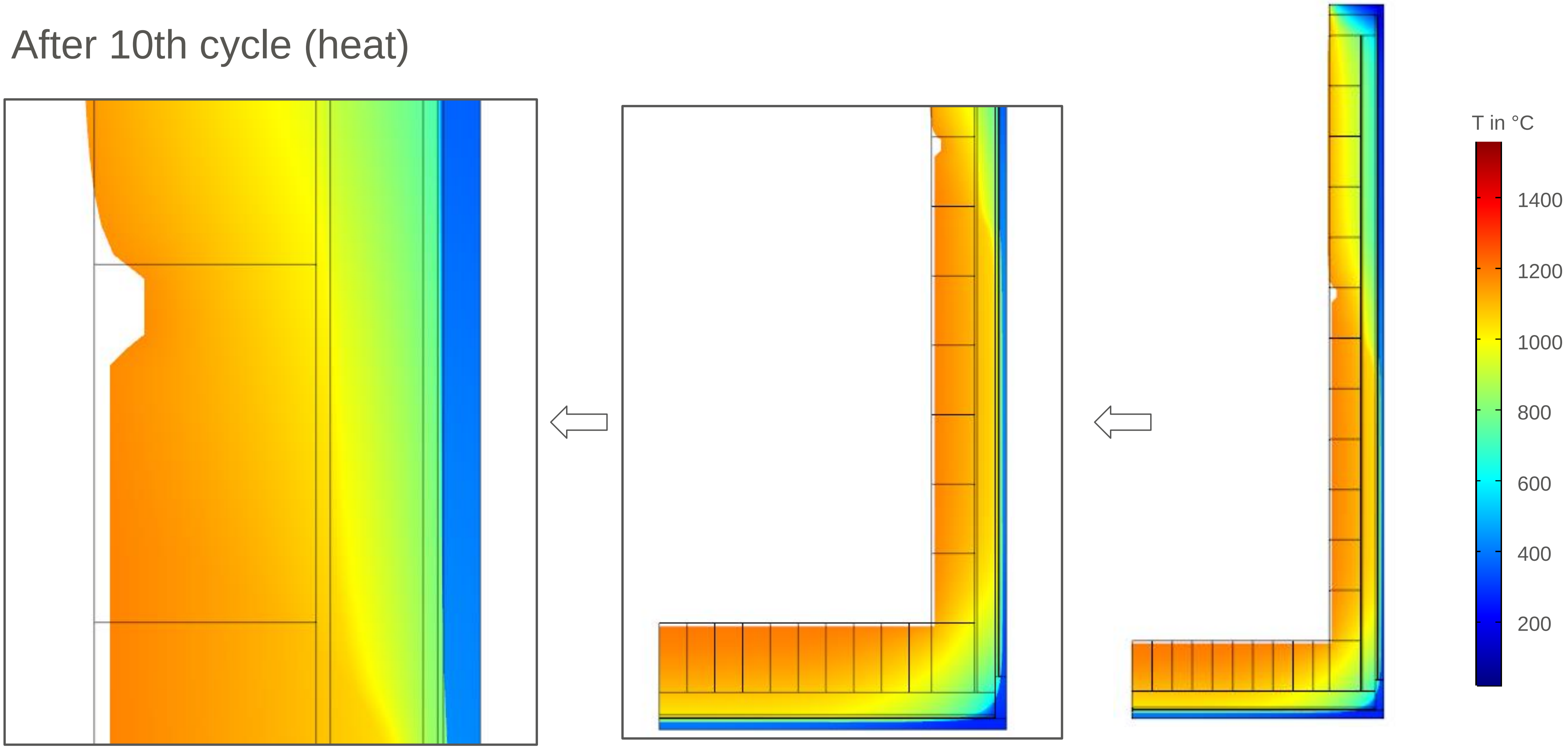
Results – Temperature distribution with wear

- After 1st cycle (heat)



Results – Temperature distribution with wear

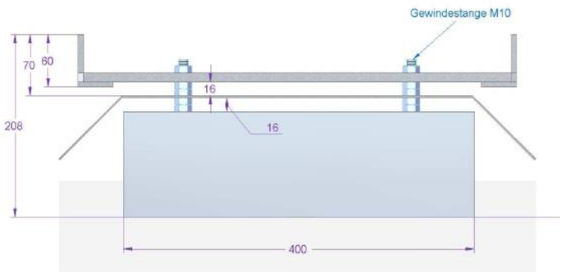
- After 10th cycle (heat)



Validation

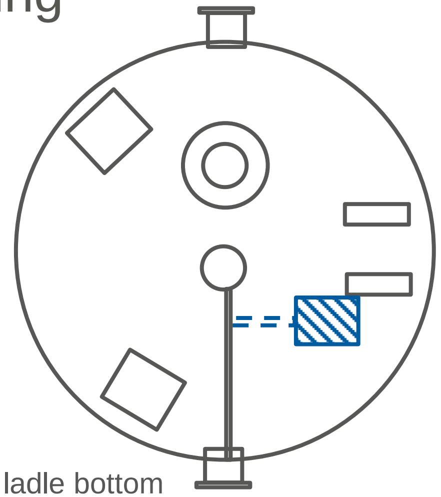
For validation of the FEM model, temperature measurements in the refractory lining of SWG ladle are planned. The results will then be compared to the results of the FEM model. Following this, the parameters in the model might be adapted if necessary.

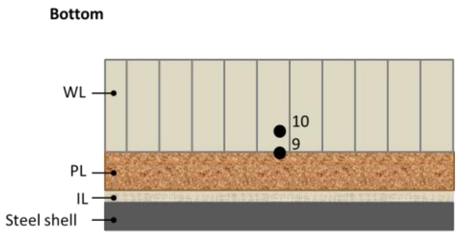
- Concept developed including protection box at bottom of SWG ladle, datalogger for 10 thermocouples and vacuum proof, thermocouples and installation paths at ladle
- Protection box designed, constructed and installed
- Temperature measurement with dummy at position of datalogger ongoing
- Selection of measurement positions ongoing

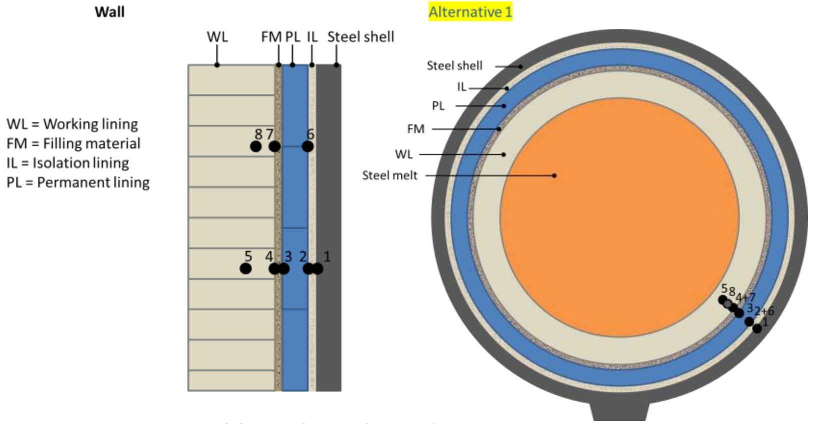


Design suggestion for protection box


Dummy for T measurement at datalogger position


Position at ladle bottom to mount protection box





Measurement positions in ladle refractory

Summary and next steps

- FEM model setup based on information from SWG
- FEM model calculates temperature and stress distribution in refractory and steel shell
- FEM model adapted to consider wear (by displacement) and changing refractory conditions (by parametrised material properties as far as available)
- Model ready to be validated and to calculate different scenarios (e.g. changing lining concept)

Next steps:

- Based on results of T measurement in box – adaption of cooling if necessary
- Order of thermocouples
- Measurement campaign
- Adaption of FEM model if necessary