Cutting with geometrically undefined cutting edges

Simulation Techniques in Manufacturing Technology

Lecture 10

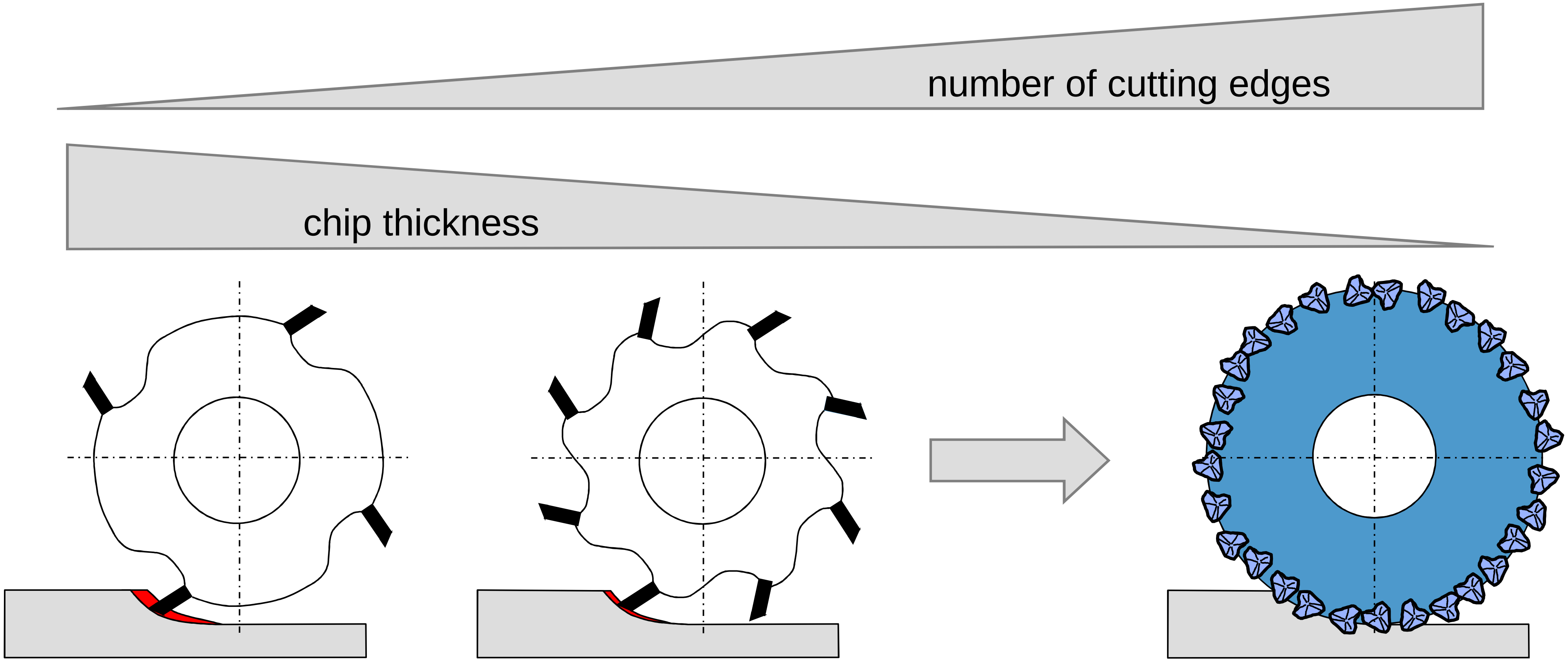
Laboratory for Machine Tools and Production Engineering

Chair of Manufacturing Technology

Prof. Dr.-Ing. Dr.-Ing. E.h. Dr. h.c. Dr. h.c. F. Klocke

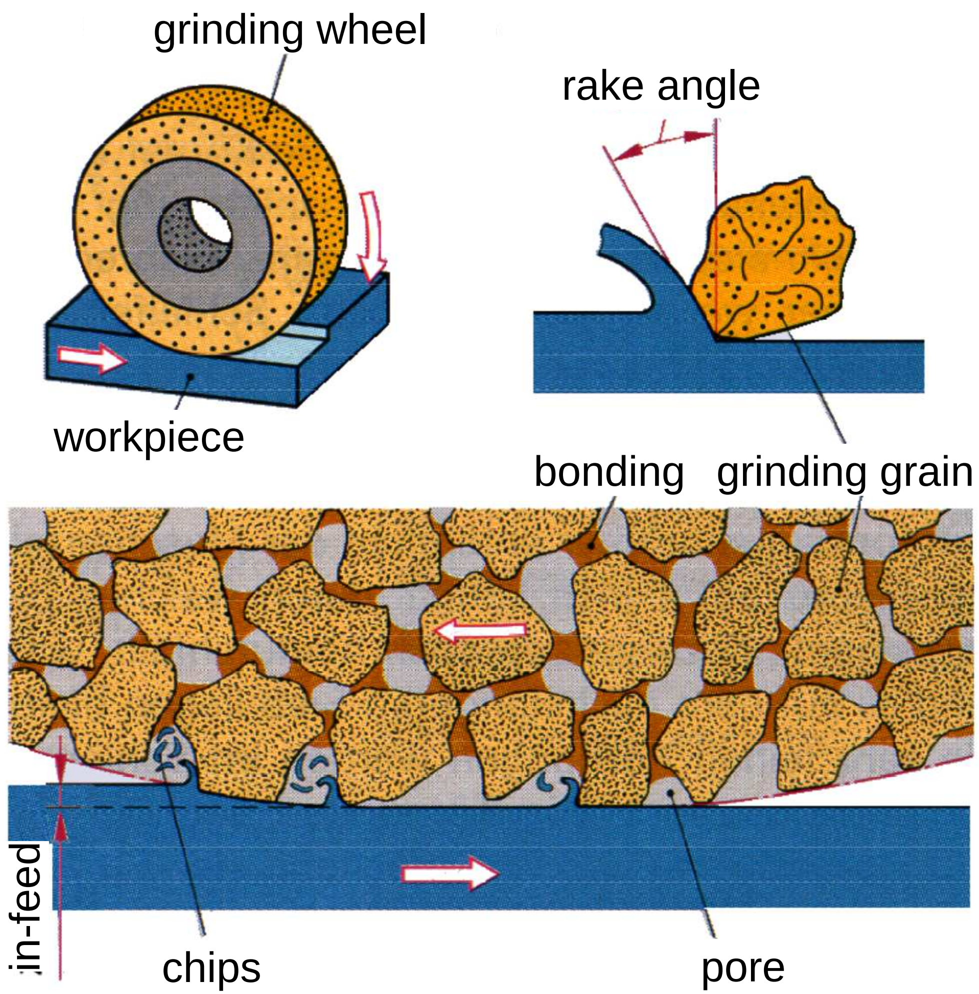
Agenda

Review:
From milling to grinding

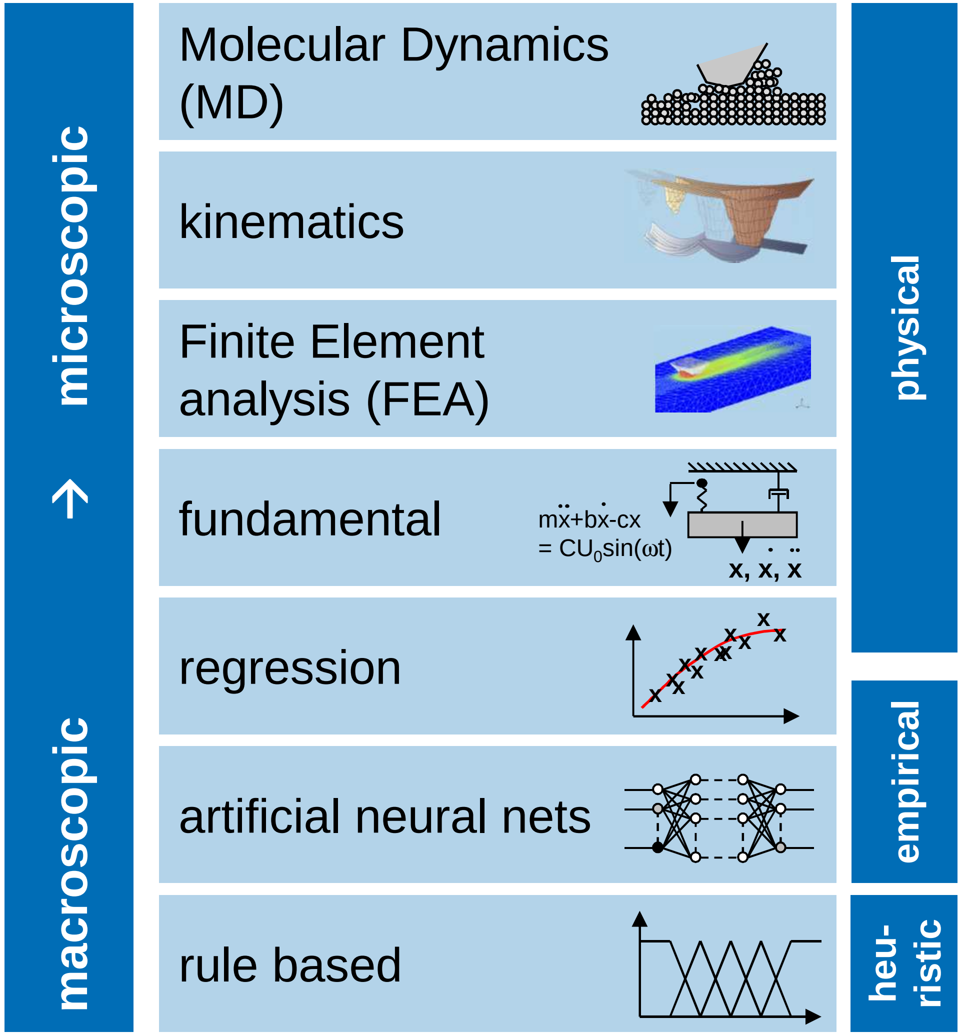


Review:
Characteristics of grinding

- cutting edges possess different geometries
- mainly highly negative chip angle
- varying distance of the cutting edges and thus different chip thicknesses
- varying distance of the cutting edges from the rotation axes
- tool consists of three components (grain, bonding, pore)
- tool can be dressed in the machine



Review:
Modelling and simulation of grinding processes



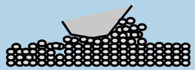
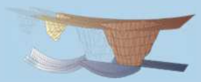
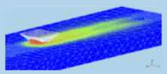
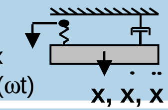
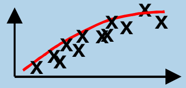
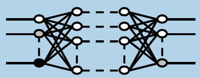
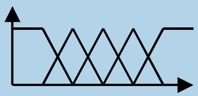
- heuristic and empirical models are limited and difficult to transfer from one process to another
- Finite Element models are complex to apply and the necessary material properties are often not known
- Molecular dynamics are very fundamental
- fundamental models can be regression models with physical background
- kinematics models can be used for applicable simulations

Source: CIRP Keynote Paper 2006, Brinksmeier et al.

Fraunhofer IPT   WZL RWTH AACHEN

Review:
Comparison of model types

● highly
◐ medium
○ low

	starting effort	CPU needed	knowledge needed	maintenance + development	amount of data	effort for experiments	effort for data analysis	transferability to other processes
Molecular Dynamics (MD) 	●	●	●	○	○	●	●	●
kinematics 	○	●	●	◐	◐	◐	●	●
Finite Element analysis (FEA) 	●	●	●	◐	○	●	●	●
fundamental $m\ddot{x}+b\dot{x}-cx = CU_0\sin(\omega t)$ 	●	○	●	●	◐	●	○	○
regression 	◐	○	○	○	●	●	○	○
artificial neural nets 	◐	○	○	○	●	●	○	○
rule based 	●	○	●	●	◐	●	○	◐

Source: CIRP Keynote Paper 2006, Brinksmeier et al.

Fraunhofer IPT

WZL RWTHAACHEN

Review:
Difficulties in grinding process simulation

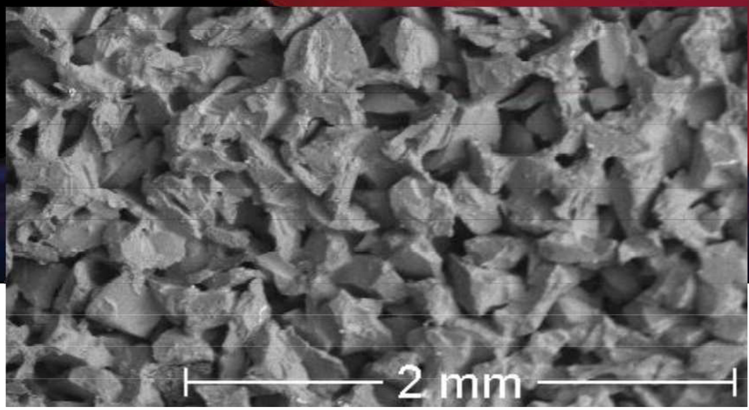
Cutting speeds:
$v_c \approx 15 - 200$ m/s

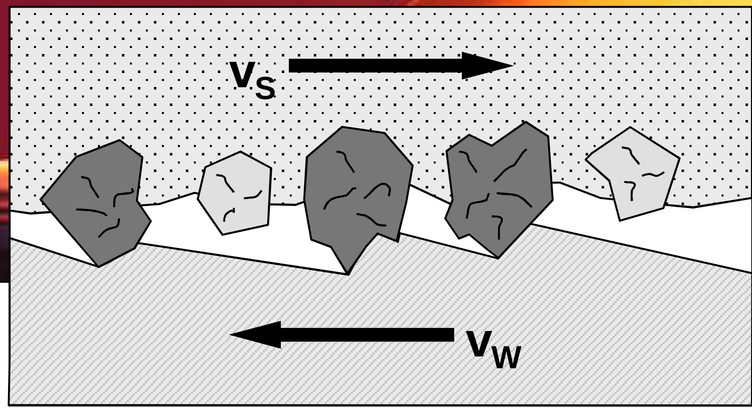
Temperatures:
peaks above 1200 °C

Temperature gradients:
10^6 °C/s / 10^3 °C/mm

Forming speeds:
$\dot{\varphi} \approx$ up to 10^7 1/s

Many material properties are not known within these ranges





Agenda

1 Repitition of lecture 9

2 Thermal heat flux in grinding

3 Force and temperature measurement

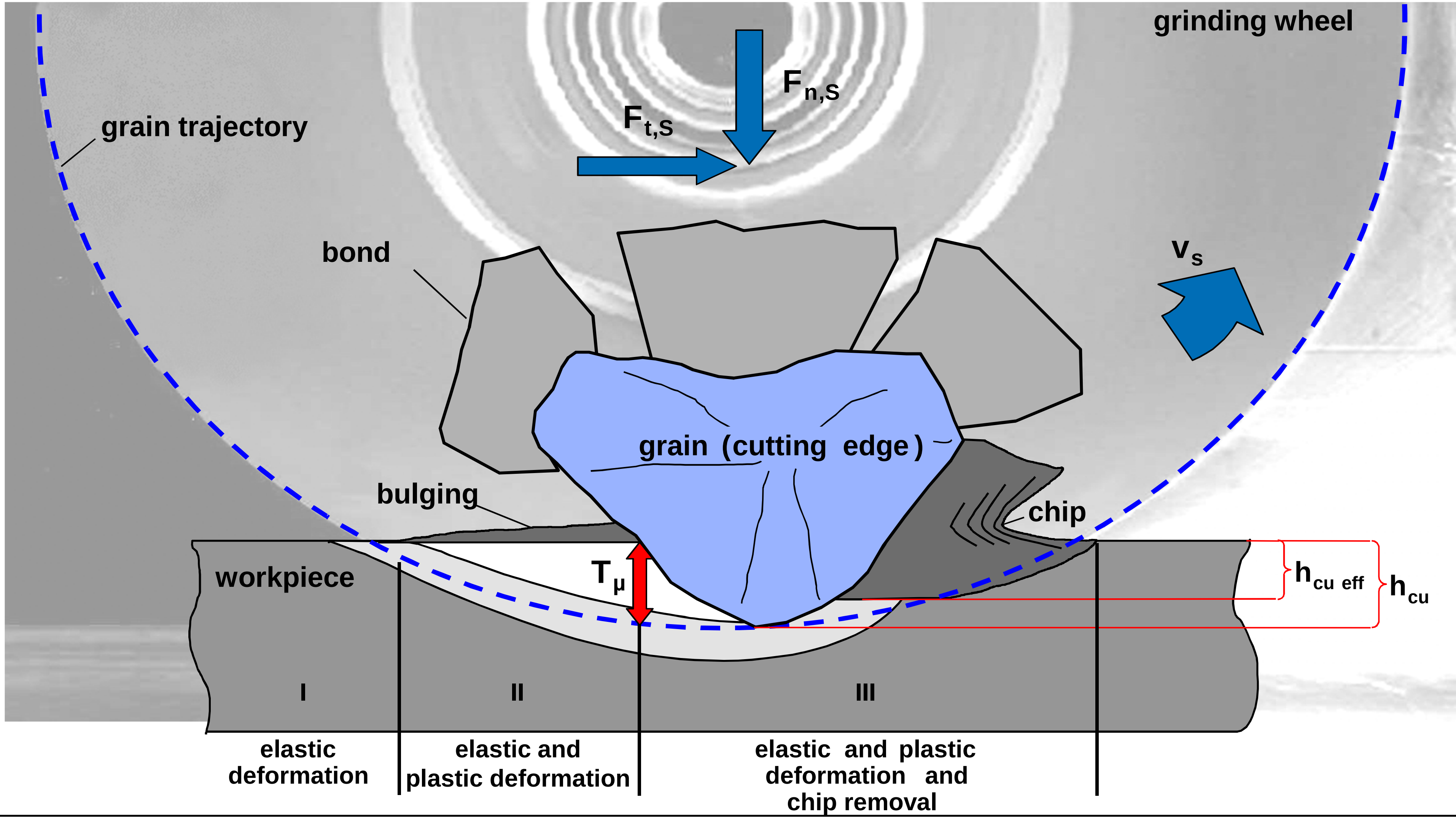
4 Practical investigation

5 FEM simulation for surface grinding

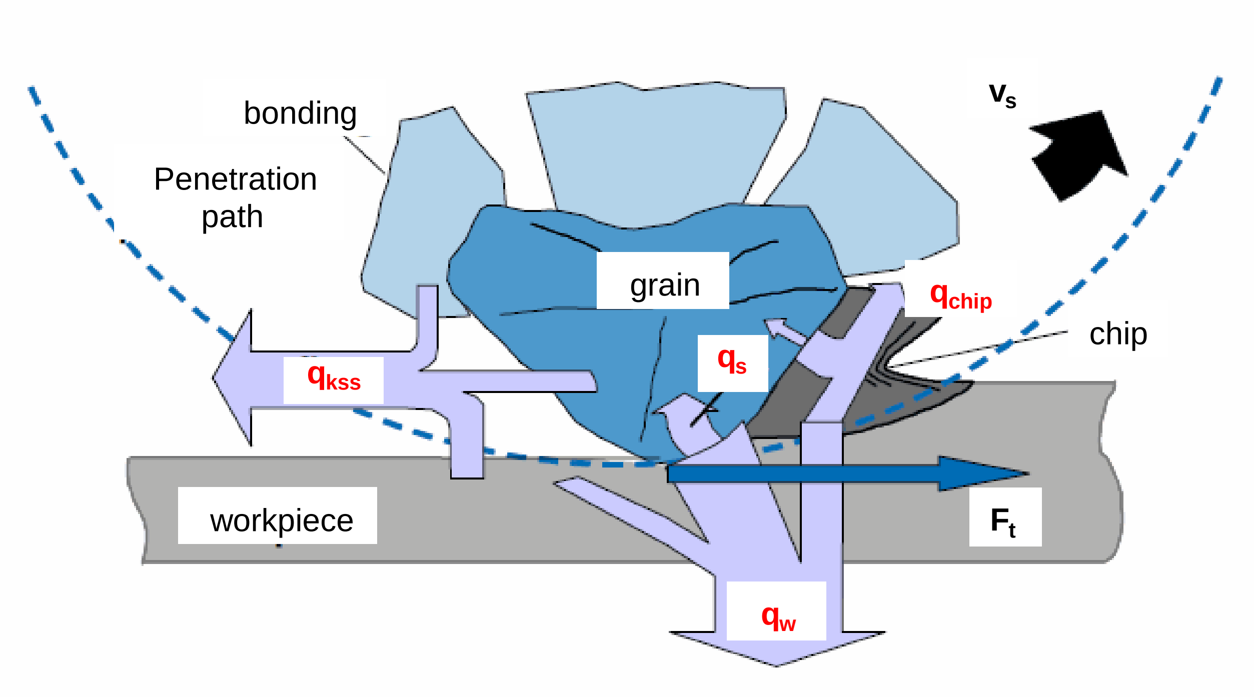
6 FEM simulation for cylindrical grinding

7 Attachement

The grinding process - Chip formation in grinding



Energy distribution and heat flow



$$q_t = q_{cool} + q_{chip} + q_w + q_s = P_c'' = F_t \cdot v_c / A_k$$

cooling lubricant — chip — workpiece — grinding wheel

$q_{kss} = R_{kss} \cdot q_t$ $\quad q_{chip} = R_{chip} \cdot q_t$ $\quad q_w = R_w \cdot q_t$ $\quad q_s = R_s \cdot q_t$

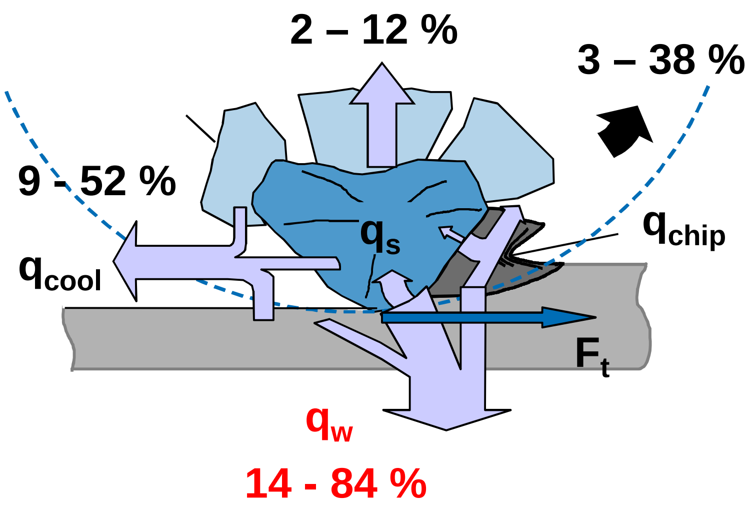
- Thermal energy flows in all relevant components of system:
  - Workpiece (q_w)
  - Grinding wheel (q_s)
  - Chip (q_{chip})
  - Cooling lubricant (q_{Cool})
- The distribution of the heat flow can be manipulated

  Legend:
  - P_c'' = cutting power
  - F_t = tangential force
  - v_c = cutting speed
  - A_k = contact area
  - q = heat flow

Fraunhofer IPT

WZL RWTH AACHEN

Calculation of heat flux into workpiece

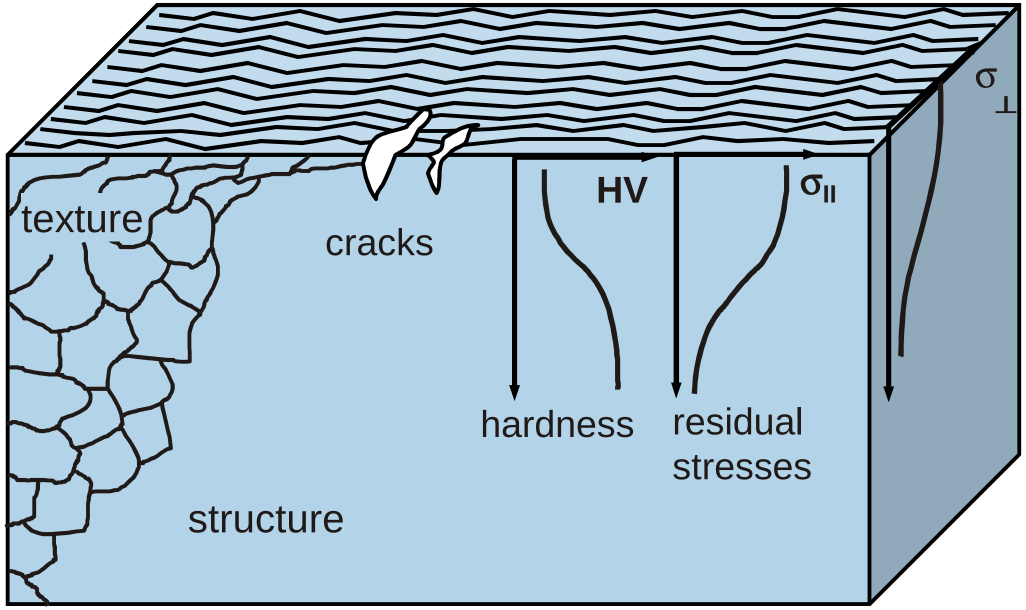


- heat flux into cooling lubricant q_{cool}
  - assumption: cooling lubricant can take heat flux until boiling point
- heat flux into chip q_{chip}
  - assumption: chips can take heat until melting point
- heat flux into grinding wheel q_s
  - grit contact analysis
  - grinding wheel contact analysis
- heat into workpiece q_w can be calculated as difference of total heat flux q_t (calculated from measured forces) and the assumed heat fluxes q_{cool}, q_{chip} and q_s

Surface integrity of the workpiece

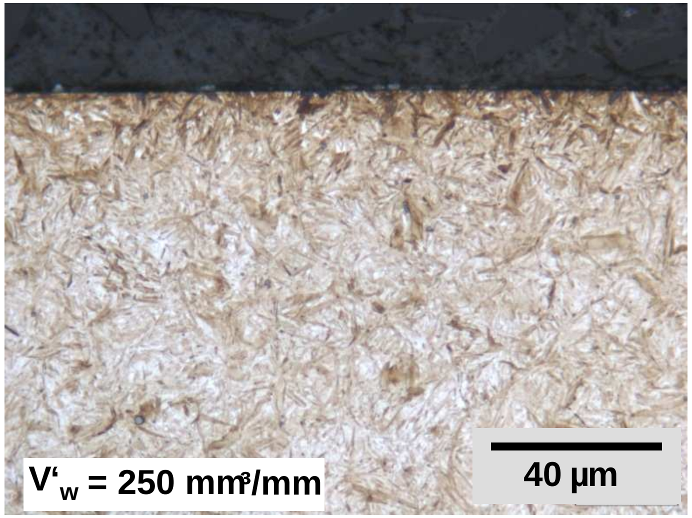
- Surface layer properties:
  - residual stresses
  - micro structure
  - micro hardness
  - roughness
  - electrical, optical, thermal, magnetical properties
  - …

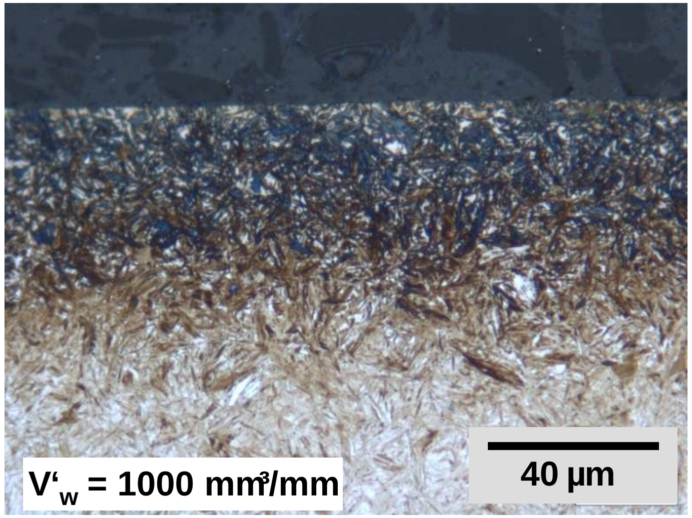
machined surface



Source: Brinksmeier, WZL

Surface integrity – Change of structure





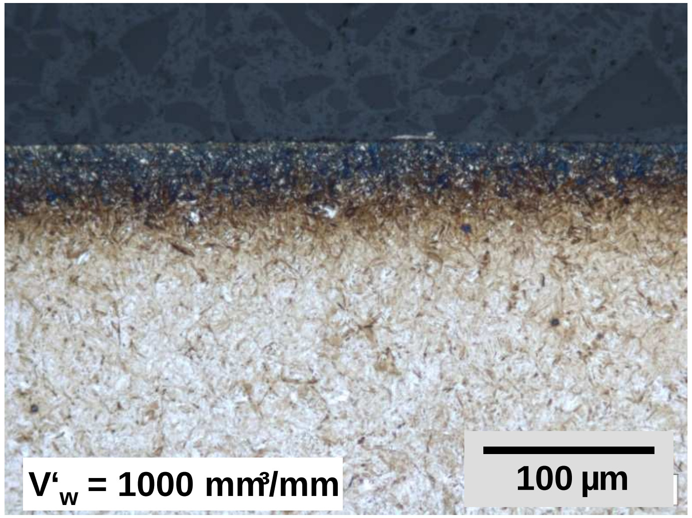
material: 16MnCr5, hard roller burnished

grinding wheel: sintered corundum A 80 H 6 V

grinding parameters:

v_c = 80 m/s; q = -120; Q'_w = 15 mm^3/mms
ext. cyl. circumferential plunge grinding

cooling lubricant emulsion 5%

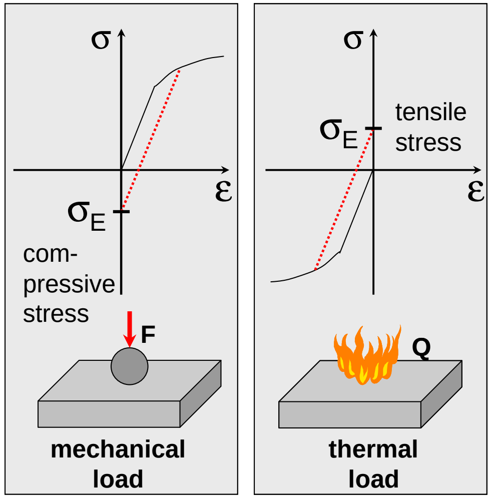


- Martensitic steels can be harmed by grinding process
  - deformation
  - rehardening at the surface possible
  - annealing in deeper regions possible
- shown case: grinding wheel wear leads to high process temperatures

Source:

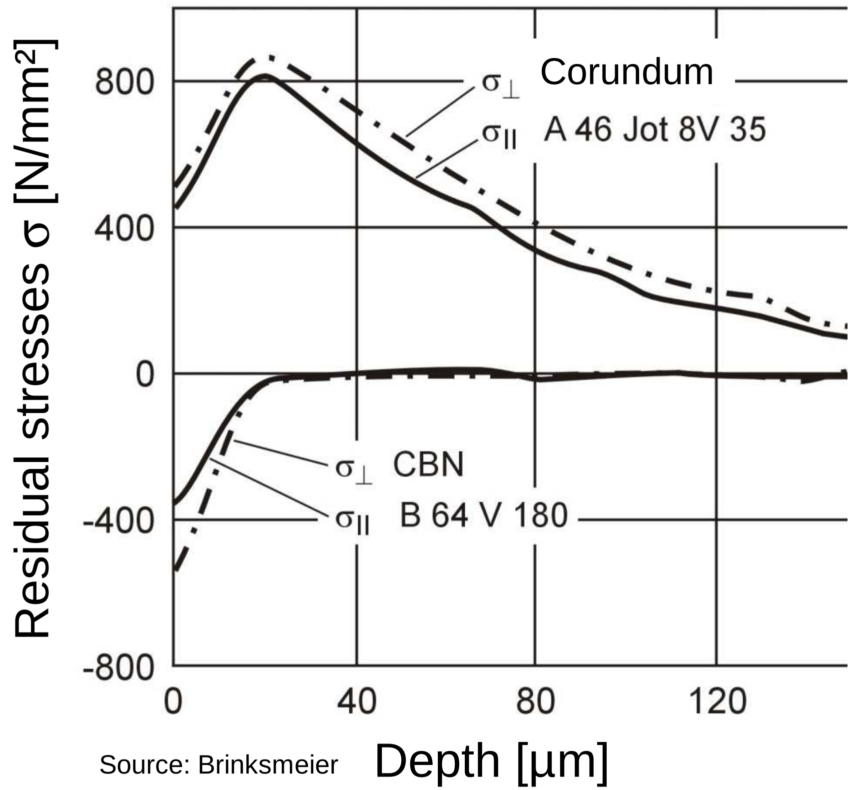
Influence of abrasive material on surface integrity

Residual stresses



- effects of thermal load during CBN machining can exceed mechanical effects
  → positive compressive stresses near surface

surface grinding	material
v_s = 30 m/s	100 Cr6V (62 HRC)
a_e = 7 µm	v_w = 400 mm/s

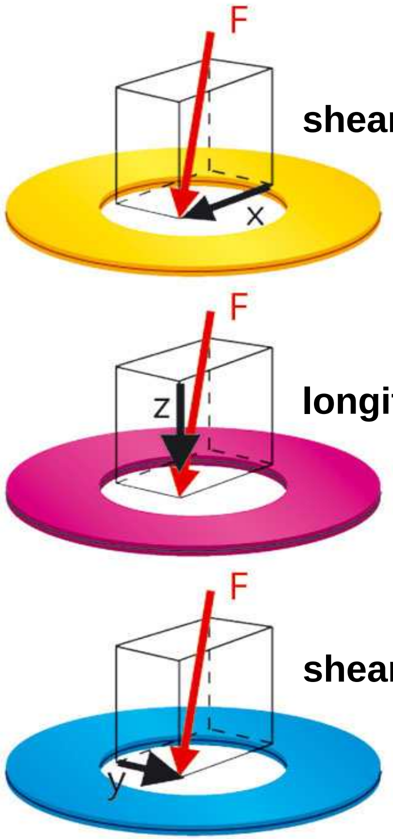


Source: Brinksmeier

Agenda

Principle of 3-component piezo-electric force measuring

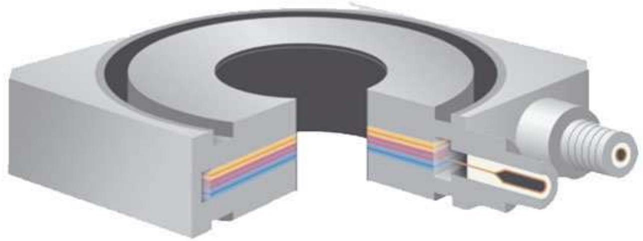
- 3 different quartzes
- measuring orientation by specific orientation of crystal axis
- integration of a charge amplifier for each component
- sealed against cooling lubricants and other fluids

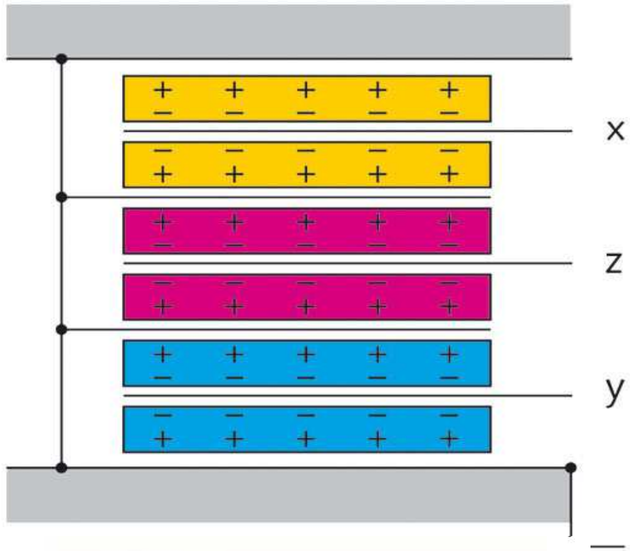


shear effect

longitudinal effect

shear effect



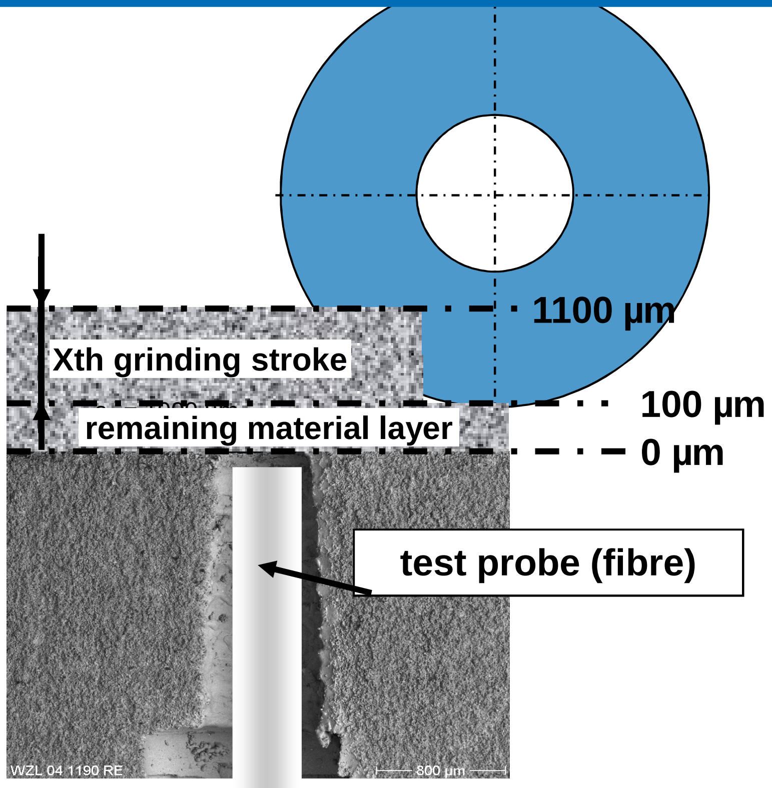


Source: Kistler

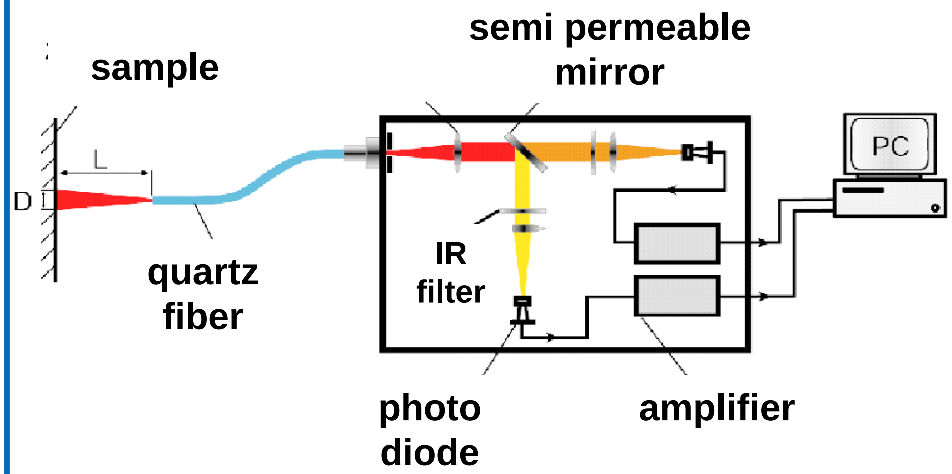
Temperature measuring methods for grinding processes

Temperature measurement method: 2-color pyrometer

test probe in workpiece

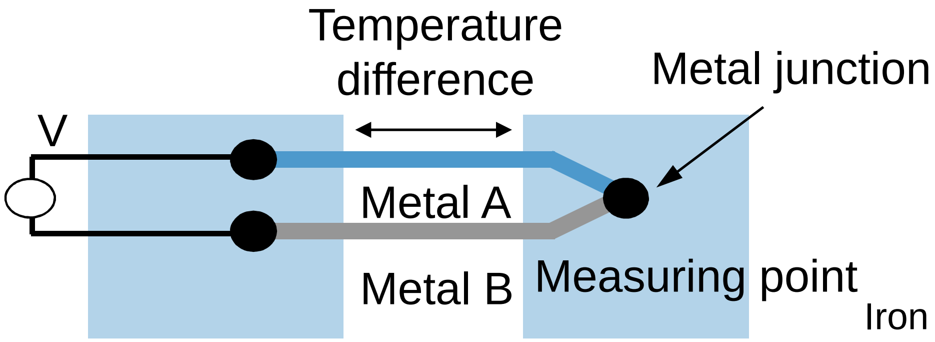


data acquisition

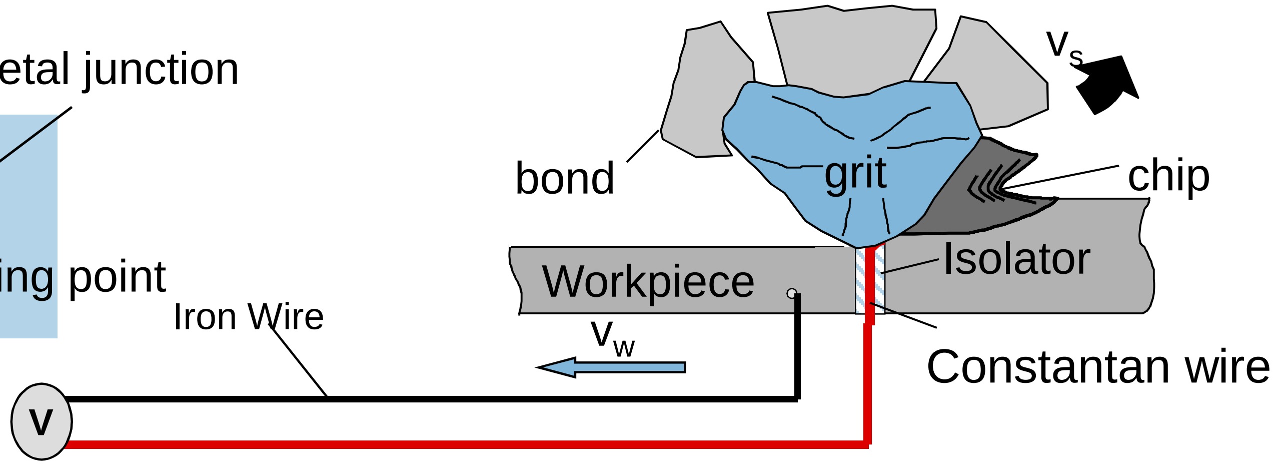


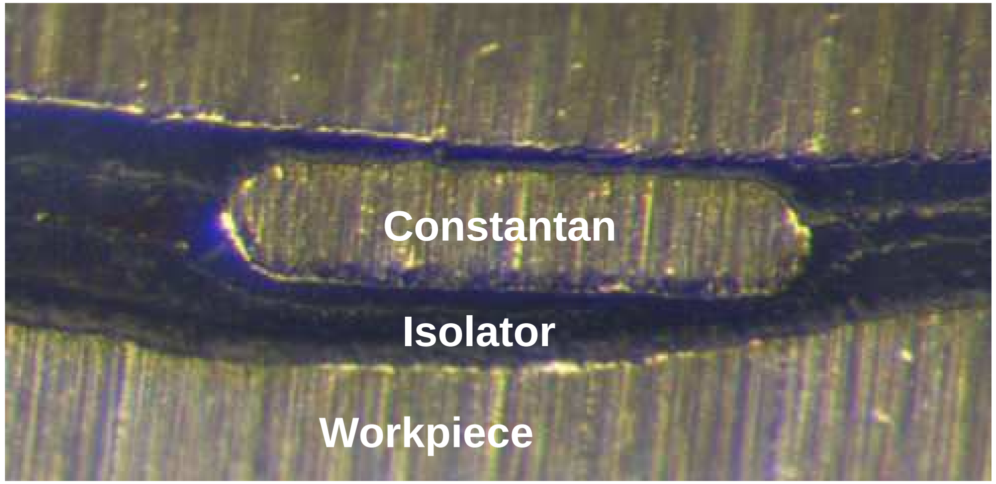
Source: WZL, Aachen

Single and double pole thermocouple measuring method



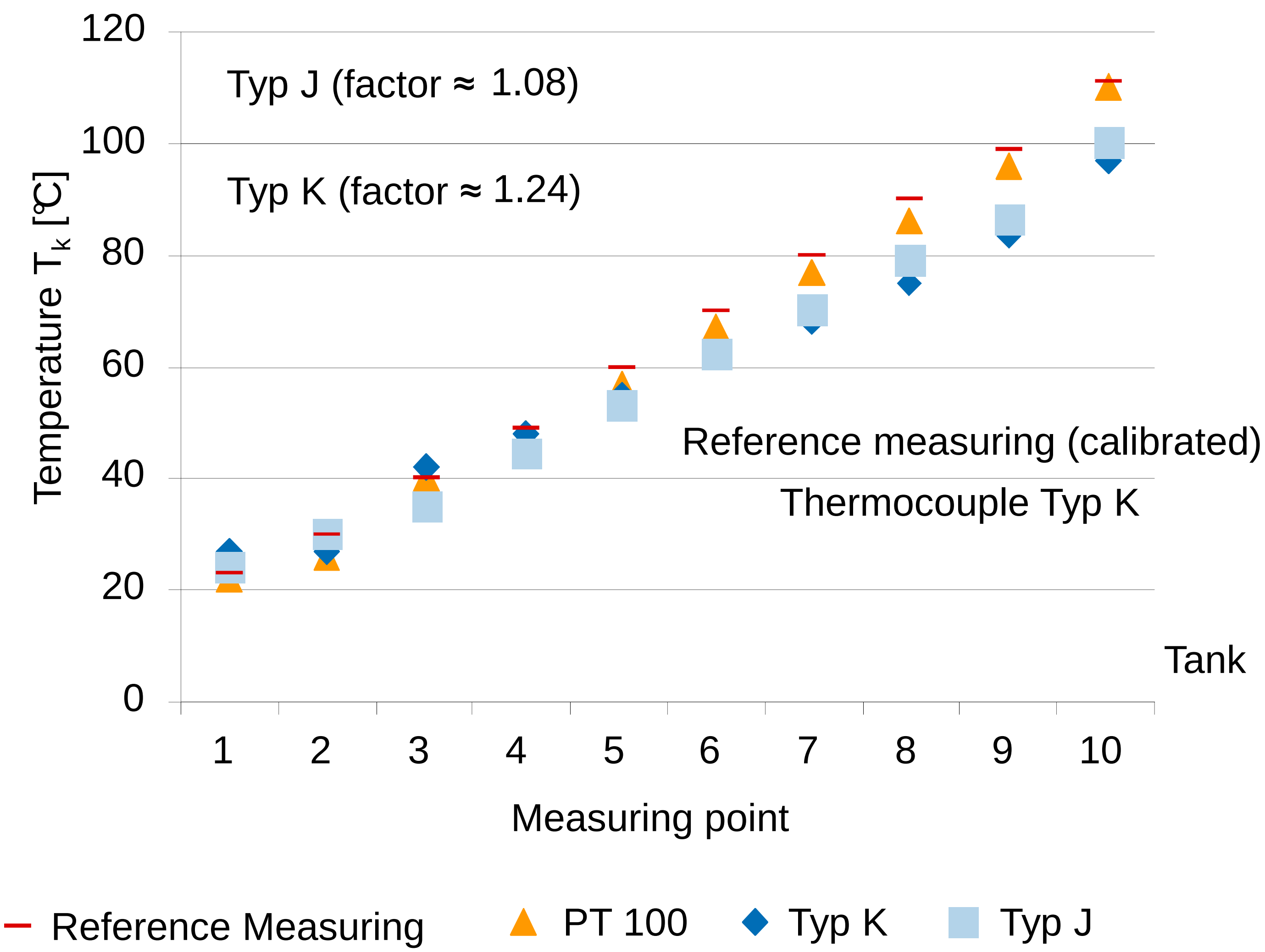
Double pole thermocouple

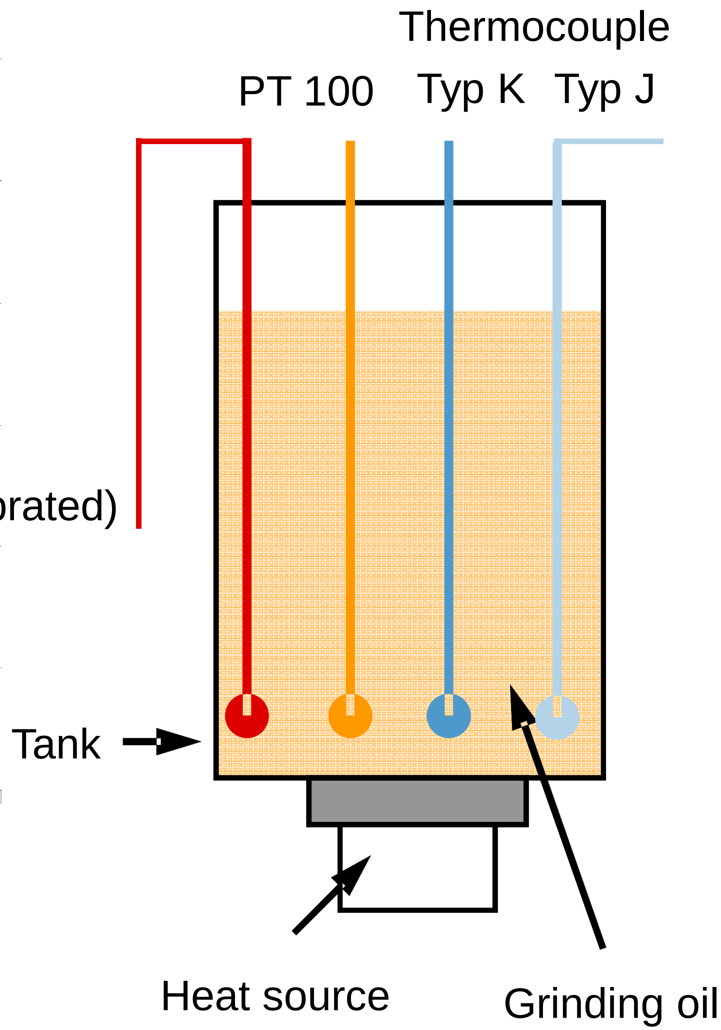




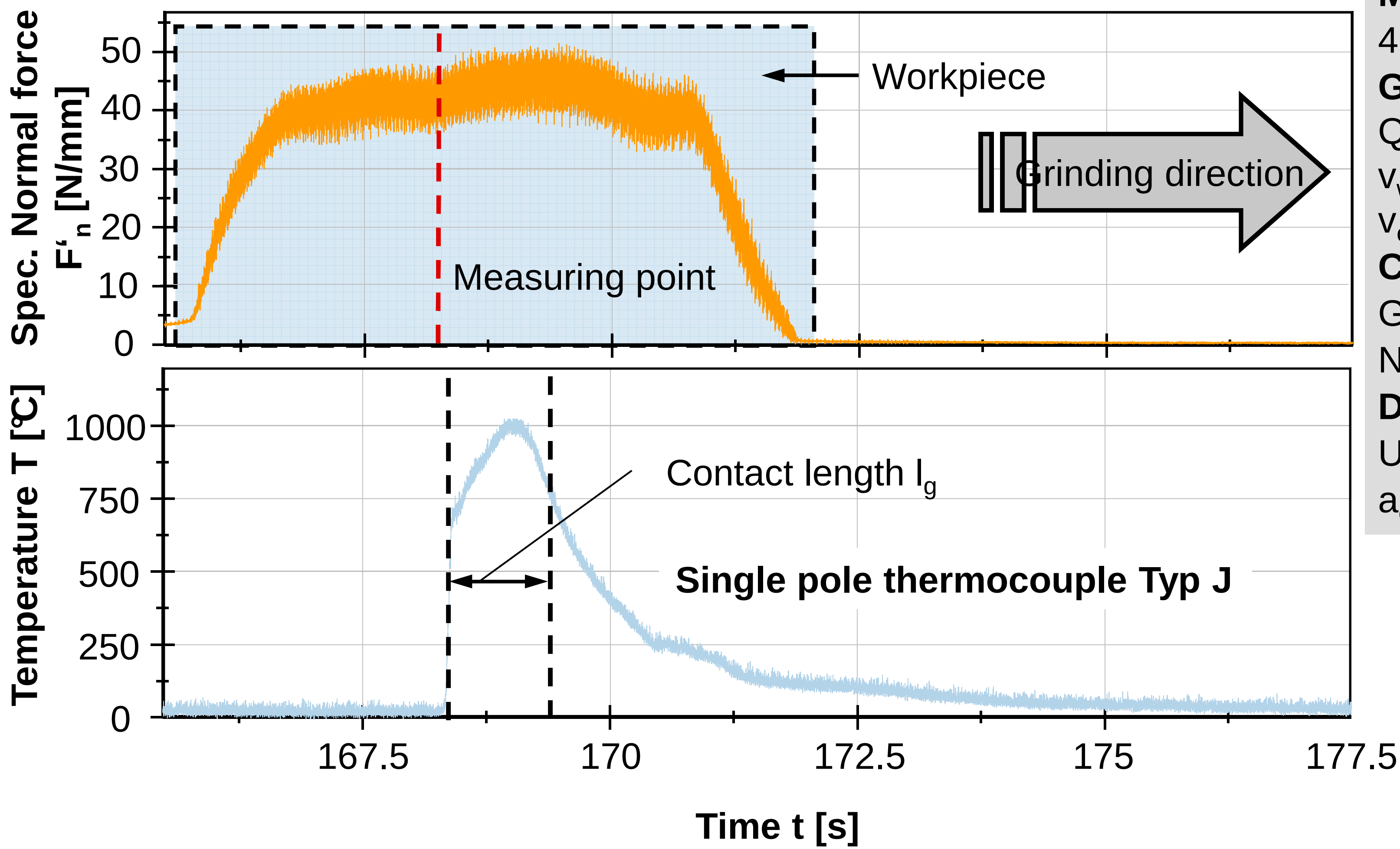
Single pole thermocouple

Calibration of different sensors





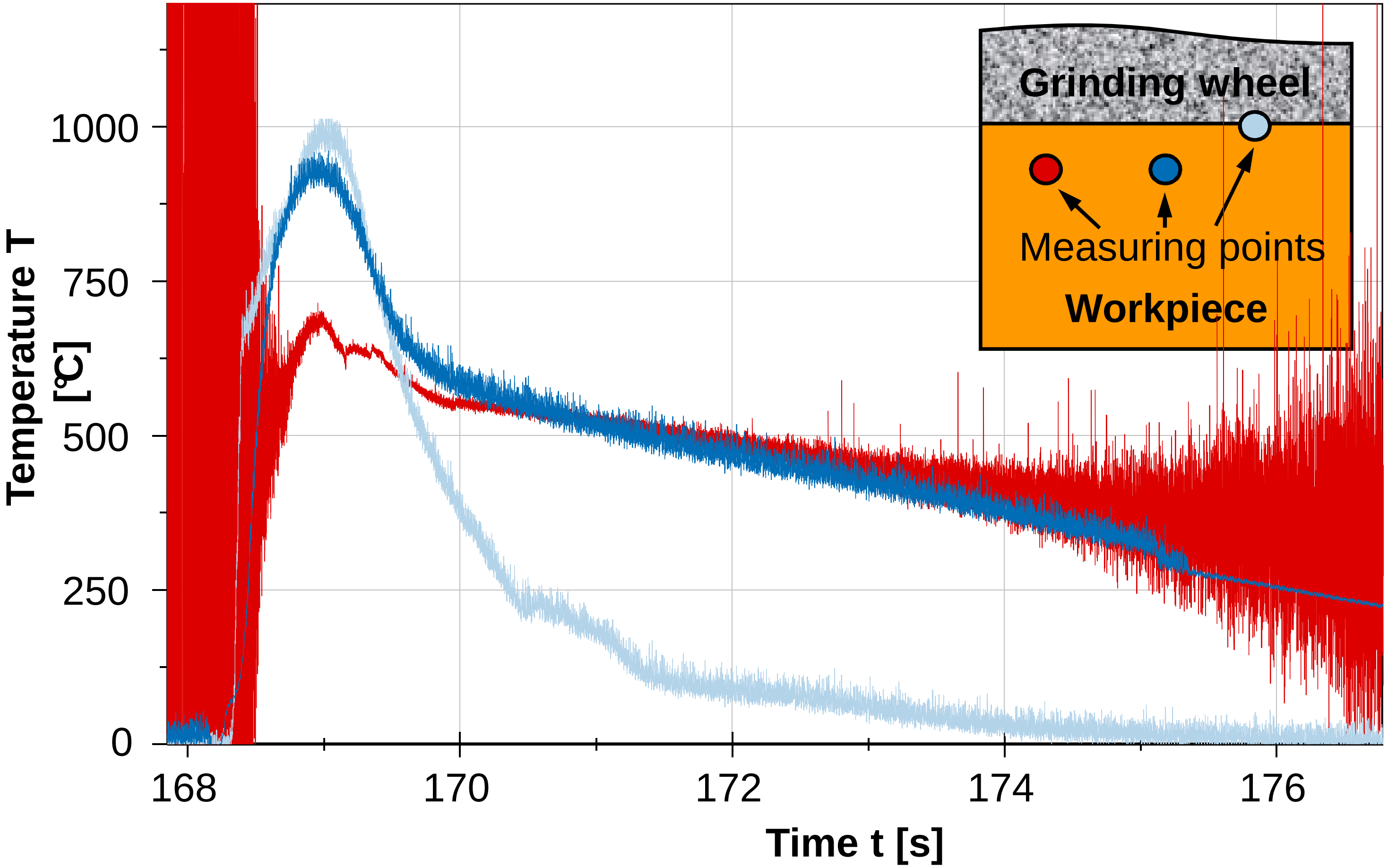
Characteristics of one grinding overrun



Material
42CrMo4
Grinding parameters
Q'_w = 2.5 mm³/mms
v_w = 600 mm/min
v_c = 30 m/s
Coolant
Grinding oil
Needle nozzle
Dressing parameters
U_d = 4
a_{ed} = 15 µm

$$l_g = \sqrt{a_e \cdot d_s}$$

Comparing the different measuring methods



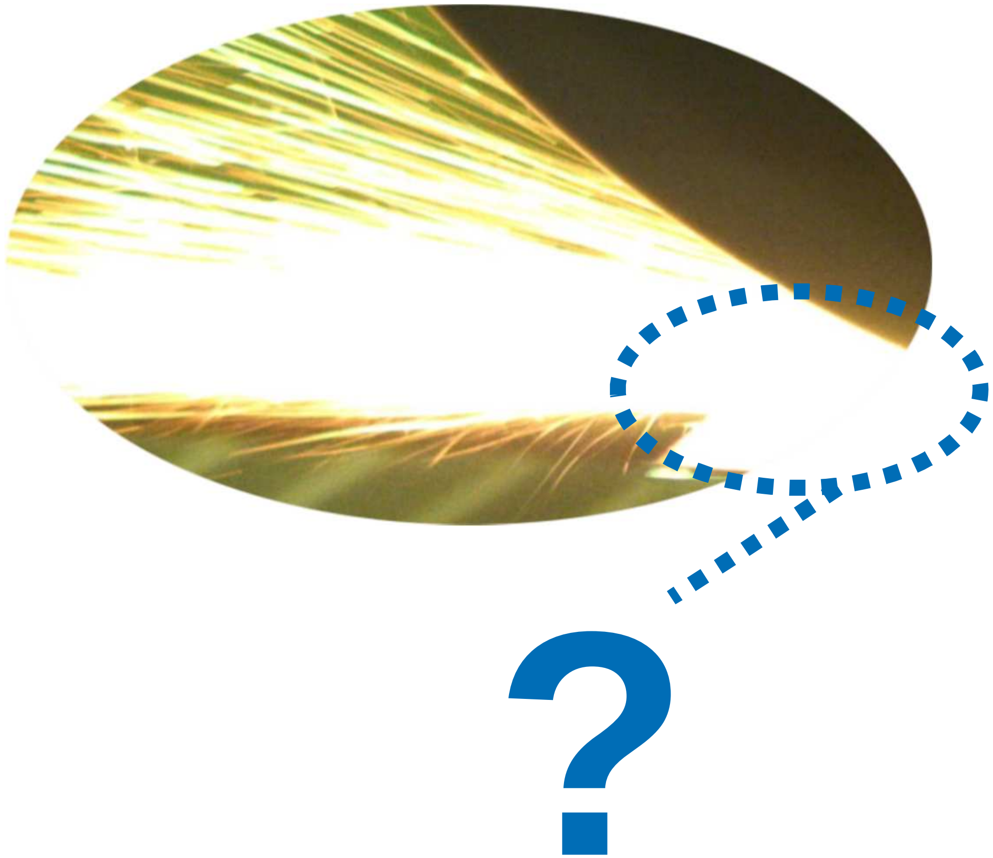
Material
42CrMo4
Grinding Parameter
Q'_w = 2.5 mm³/mms
v_w = 600 mm/min
v_c = 30 m/s
Coolant
Grinding oil
Needle nozzle
Dressing parameter
U_d = 4
a_{ed} = 15 µm

Agenda

System „Surface Grinding“

Input-Parameters

- Machine
- Workpiece
- Tool
- Preparation
- Coolant
- Process parameters
- Boundary condition



Grinding burn at the workpiece surface layer?

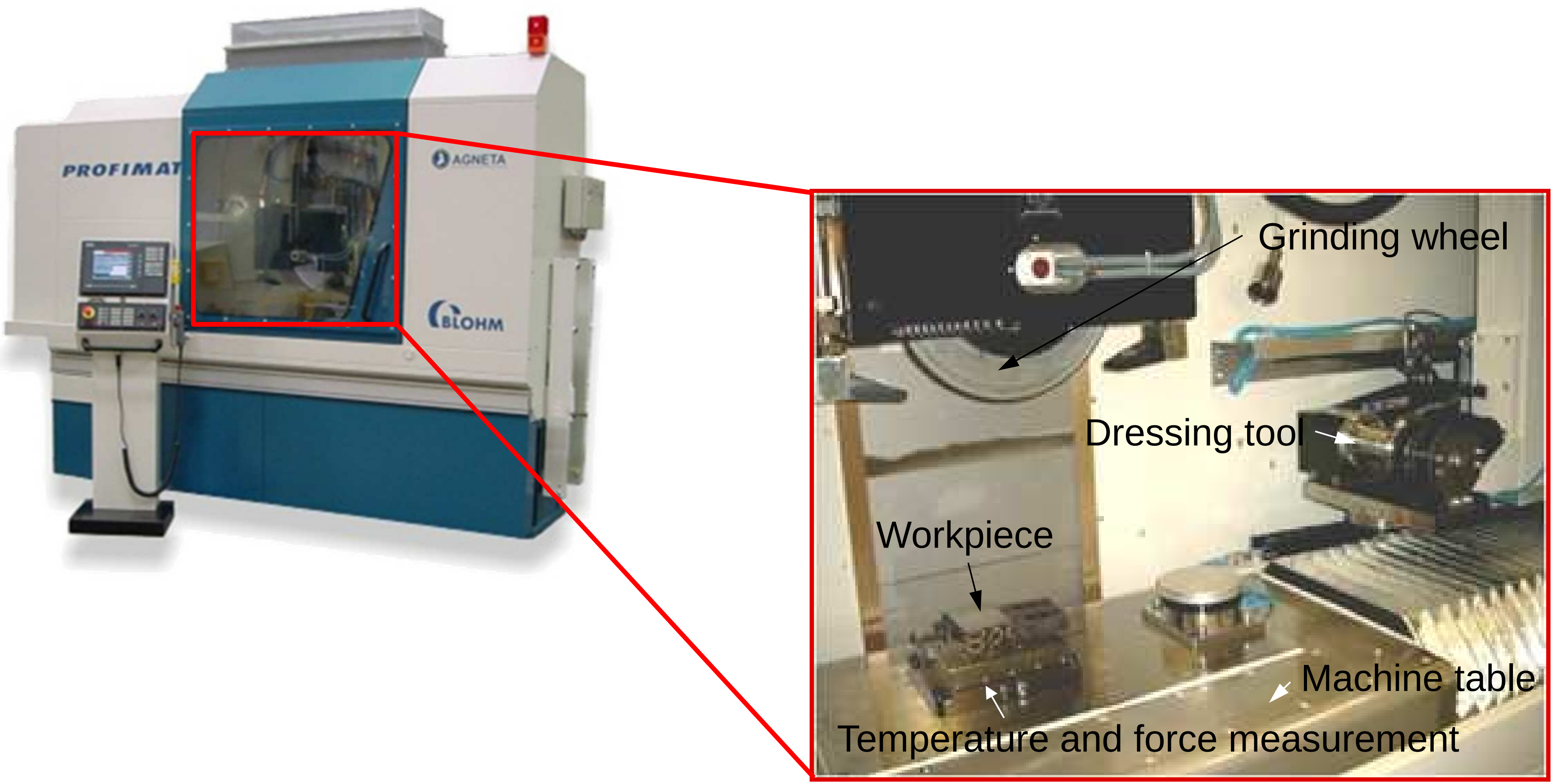
Experimental investigation in speed stroke grinding

Machine
BLOHM PROFIMAT
Material
100Cr6 (HRC 62)
Coolant lubricant
Emulsion 5%
Q_{cool} = 96 l/min

$\mathbf{P_{max}}$
40 KW
$\mathbf{d_{s,max}}$
400 mm
$\mathbf{n_{s,max}}$
11,000 min^{-1}

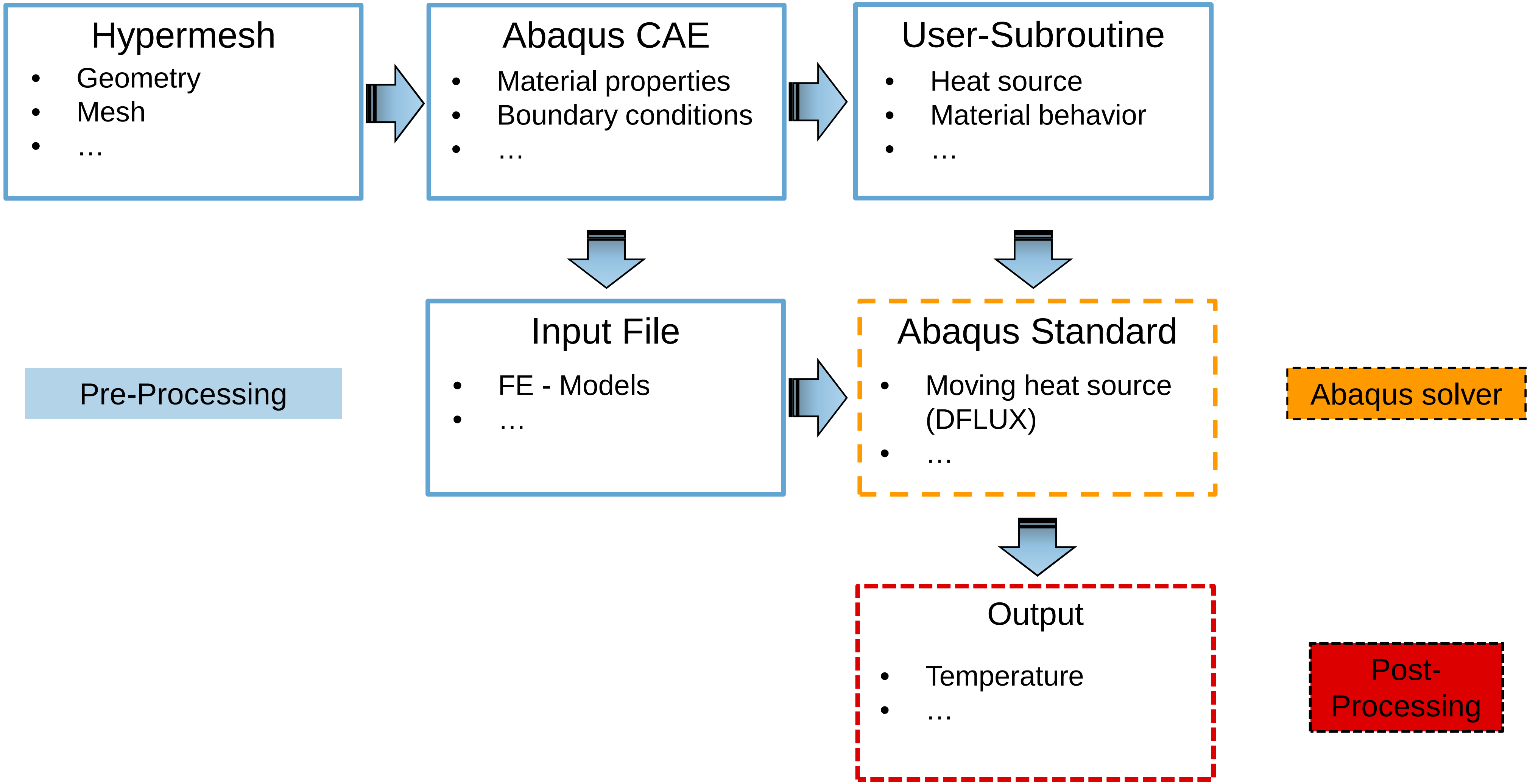
Grinding parameters				**Dressing parameters**		
Q'_w [mm³/mms]	v_w [m/min]	v_c [m/s]	V'_w [mm³/mm]	U_d	a_{ed} [µm]	q_d
10 - 45	12 - 180	80 - 160	1000	4	3	0.6 – 0.8
Grinding wheel				**Dressing tool**		
B181 LHV 160				Form roller		

BLOHM PROFIMAT 408 HT

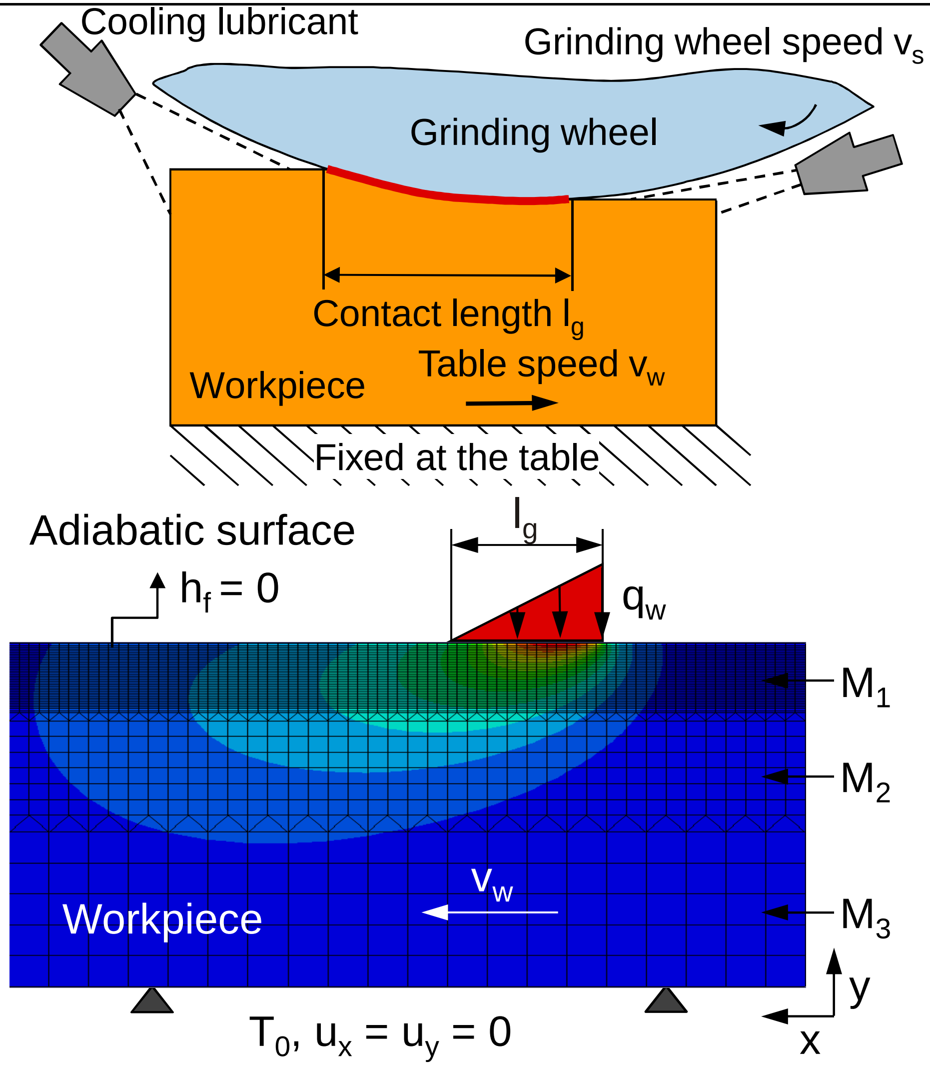


Agenda

Procedure of FE modelling processes



From the real process to a Finite Element Model

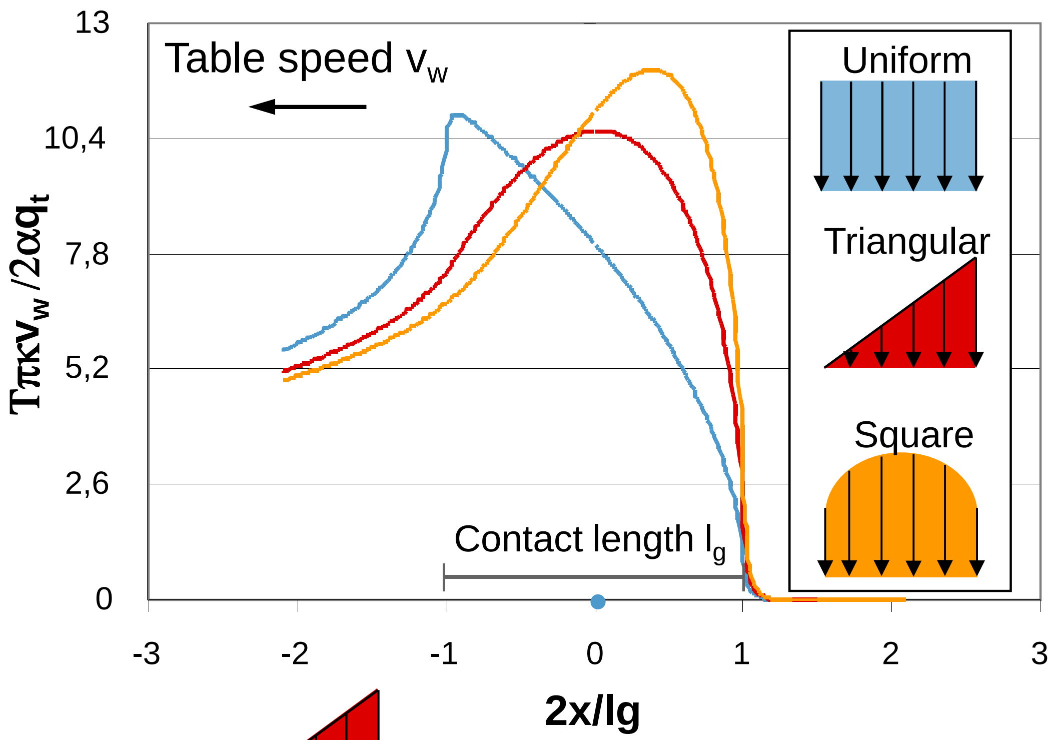


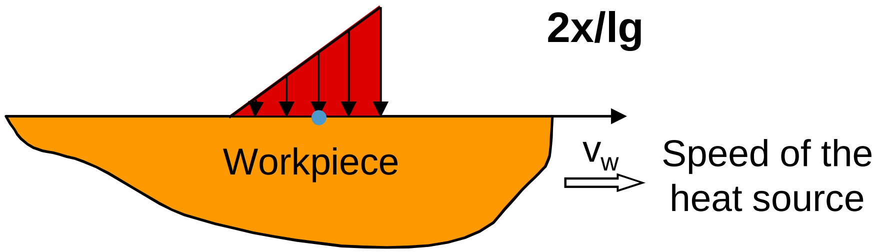
Boundaries

- Two-dimensional model
- Linear moving heat source
- Temperature-independent thermal material properties
- The surface of the solid is adiabatic
- Bottom surface is set to 20℃
- Maximum temperature of the coolant lubricant is t_B = 120℃ due to the boiling point of emulsion
- Heat flux into the workpiece
  $q_w = q_t - q_s - q_{Cool} - q_{Chip}$

In this approach of a Finite Element Model only a thermal load is considered.

Analytical calculation of different heat flux profiles in grinding



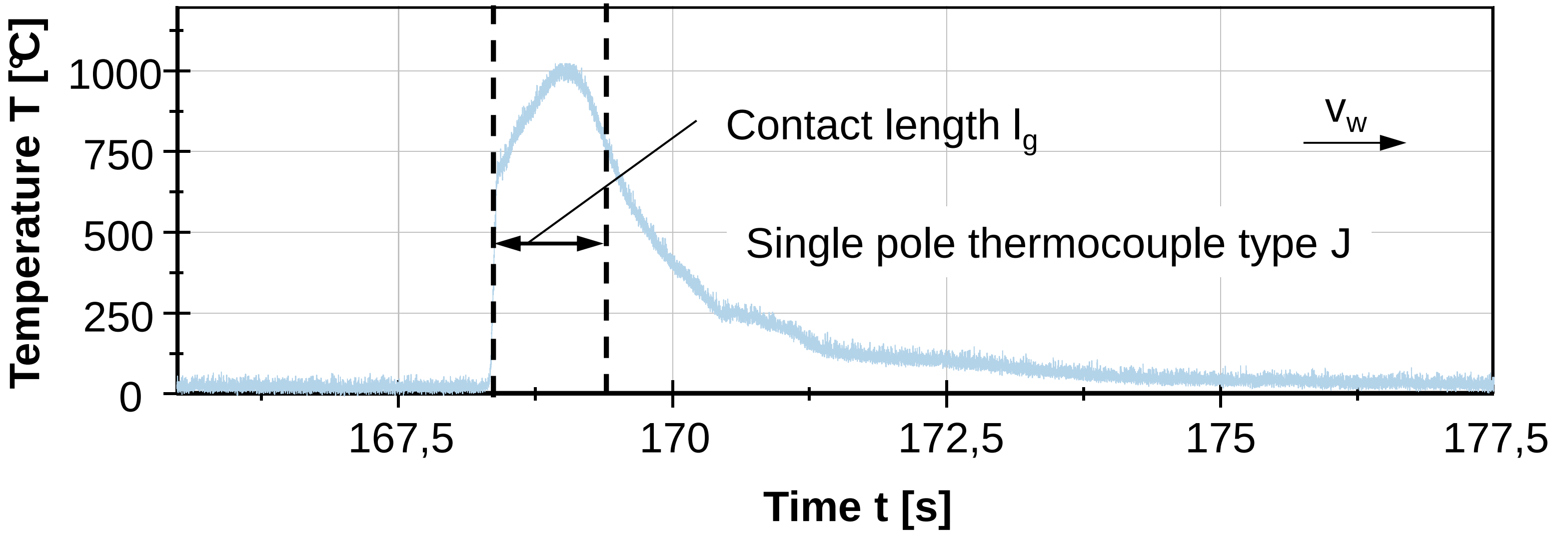


Model by Carslaw and Jaeger

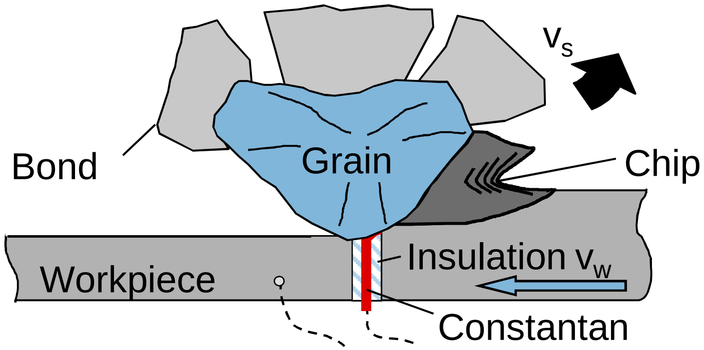
- heat source has a triangular distributed heat flow density
- heat source moves linear and with constant speed over the surface
- heat source has an unlimited expansion vertical to the direction of movement
- the heated solid is semi-infinite, i.e. it is only limited at one side

For the evaluation of the triangular heat flux profile experimental temperature measurements are necessary.

Results:
Practical investigation for heat flux profile evaluation



Material
42CrMo4
Grinding Parameter
Q'_w = 2,5 mm³/mms
v_w = 600 mm/min
v_c = 30 m/s
Coolant lubricant
Oel
Needle Nozzle
Dressing Parameter
U_d = 4
a_{ed} = 15 µm

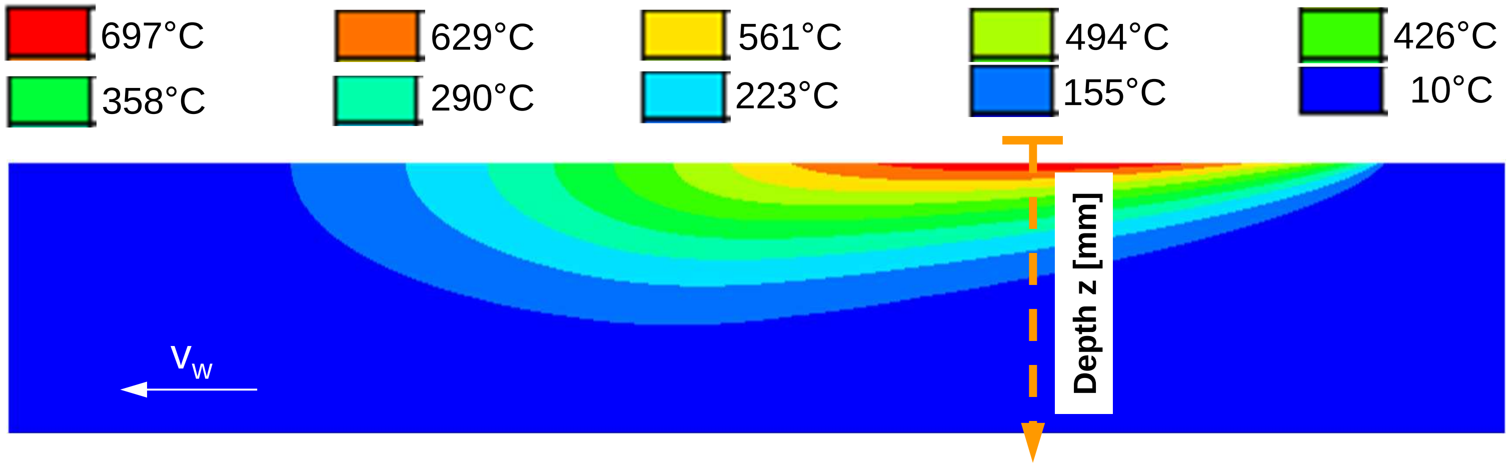


Principle single pole thermocouple type J

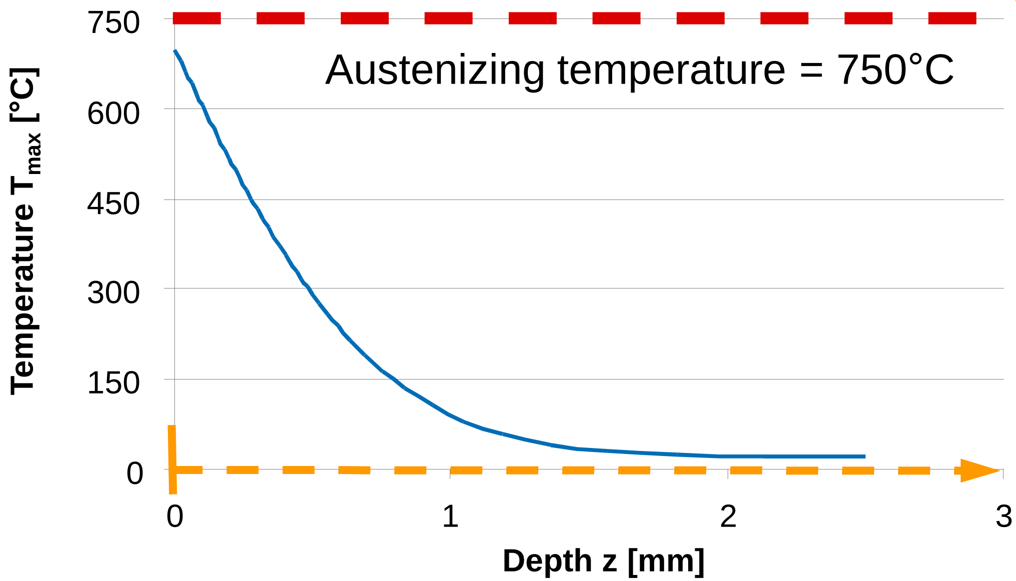
Experimental investigation showed that a triangular heat flux profile shows best results.

$$l_g = \sqrt{a_e \cdot d_s}$$

Results:
Simulation results for the creep grinding process

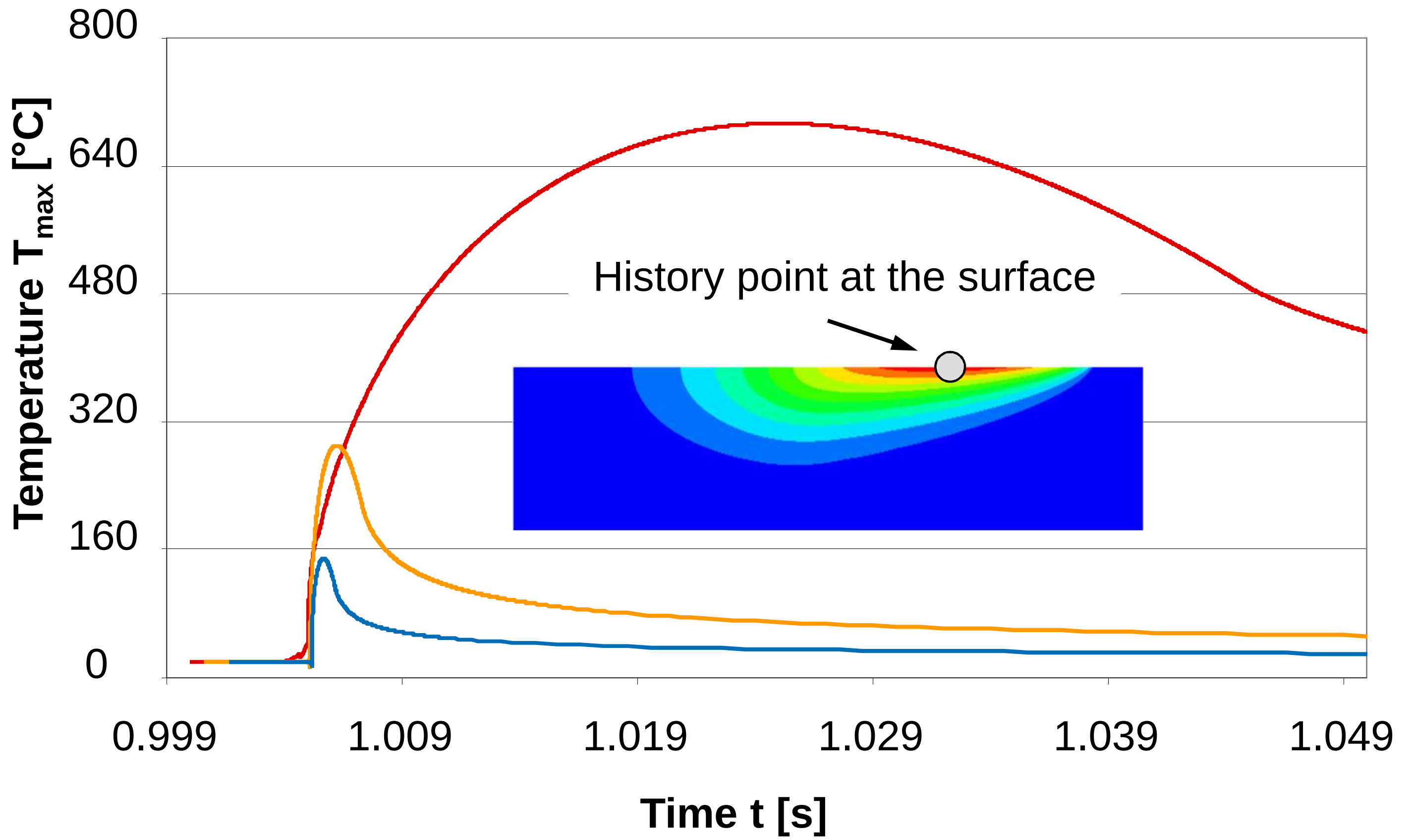


Material
100Cr6 (HRC 62)
Grinding wheel
B181 LHV 160
Grinding parameters
v_w = 12 m/min
Q'_w = 40 mm³/mms
v_s = 160 m/s
Coolant lubricant
Emulsion (5%)



- Austenizing temperature was not reached during the simulation of different grinding processes.
- Therefore, it is assumed that no phase transformation will take place.

Results:
Temperature history at the hottest point of the surface layer



Material
100Cr6 (HRC 62)
Grinding wheel
B181 LHV 160
Grinding parameters
Q'_w = 40 mm³/mms
v_s = 160 m/s
Coolant lubricant
Emulsion (5%)

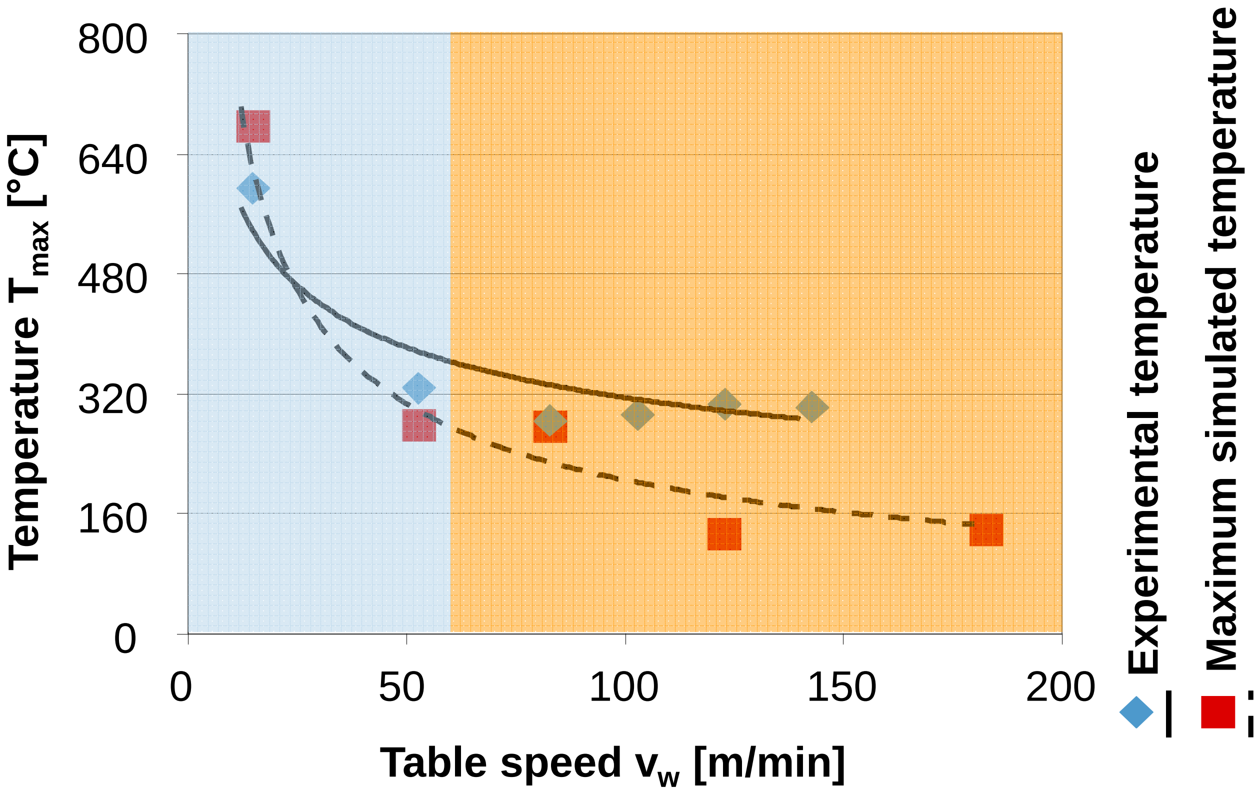
v_w = 12 m/min
v_w = 80 m/min
v_w = 120 m/min

The maximum temperature and the temperature gradient can be predicted for high table speeds.

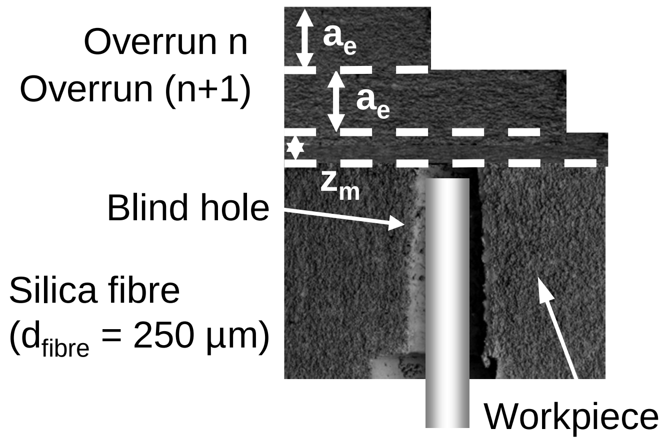
Fraunhofer IPT

WZL RWTHAACHEN

Results:
Comparison between simulated and experimental temperature



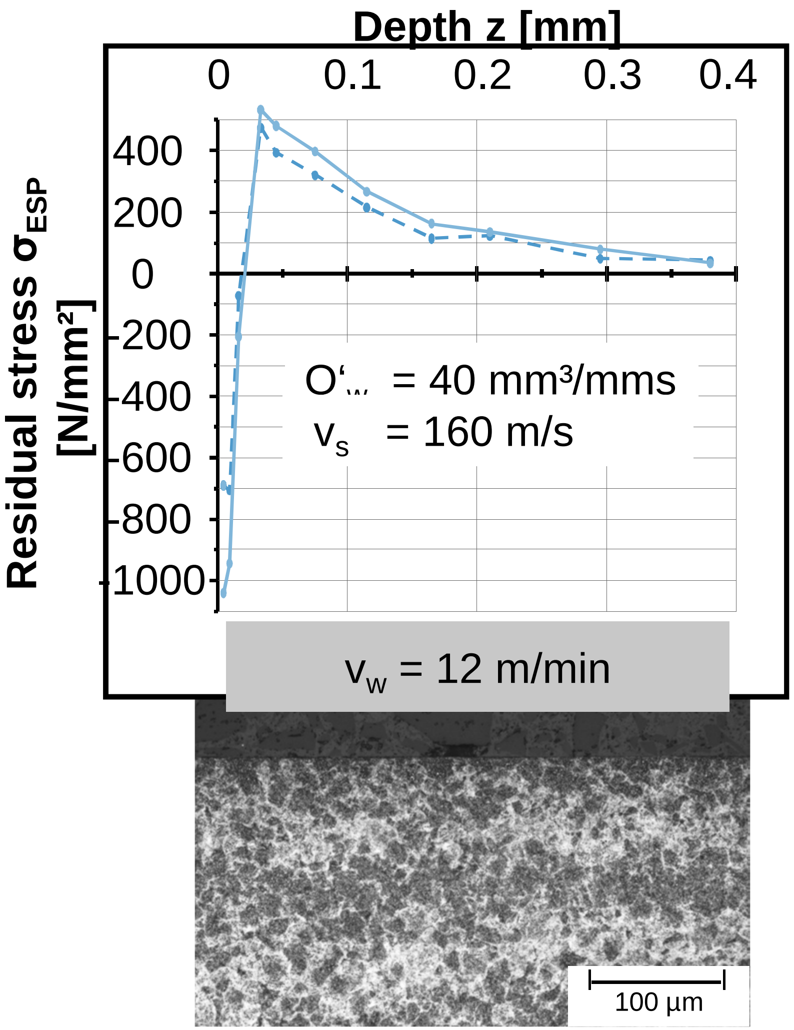
2-colour-pyrometer test set up

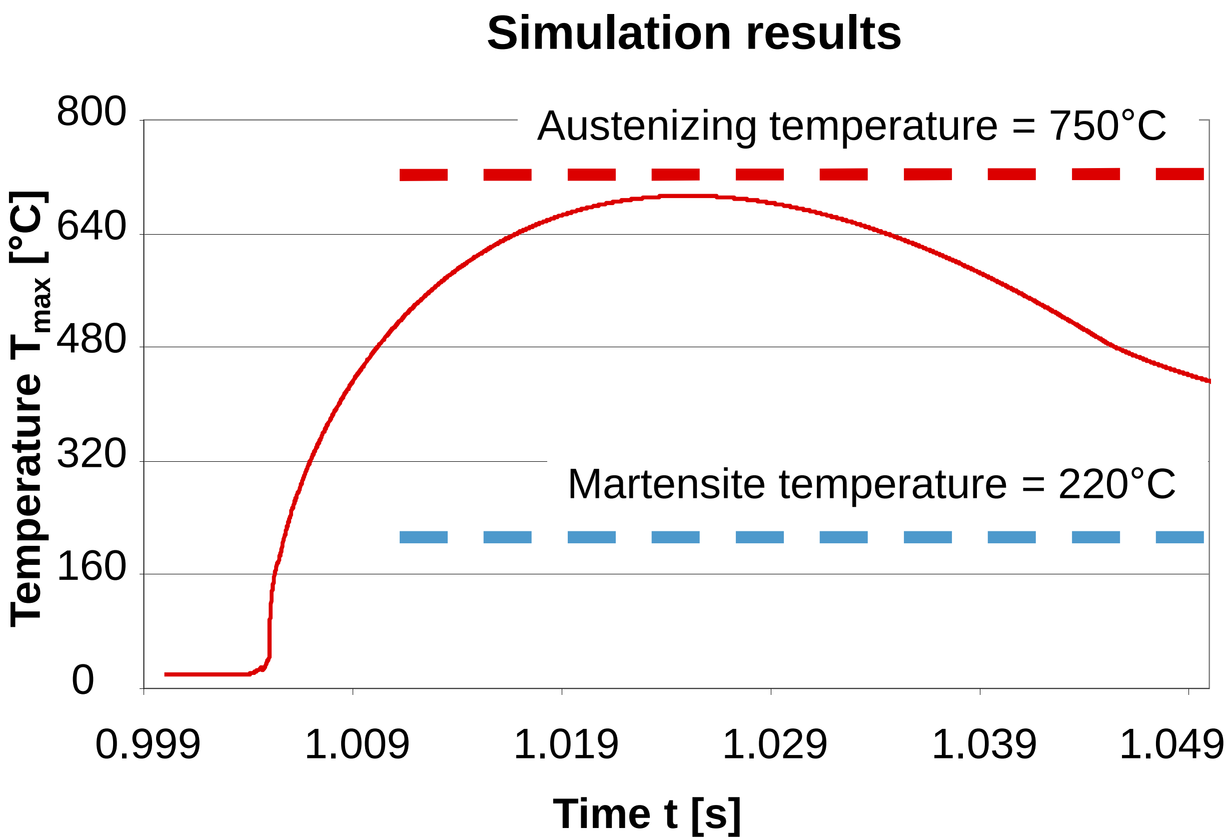


Grinding wheel	Grinding parameters	Coolant lubricant
B181 LHV 160	Q'_w = 40 mm³/mms	Emulsion (5%)
Material	v_s = 160 m/s	
100Cr6 (HRC 62)	V'_w = 1000 mm³/mm	

- For low table speeds the thermal impact has a main influence on the surface layer.
- With increasing table speeds the grinding mechanism could not be considered completely.

Results:
Validation of the FEM Simulation

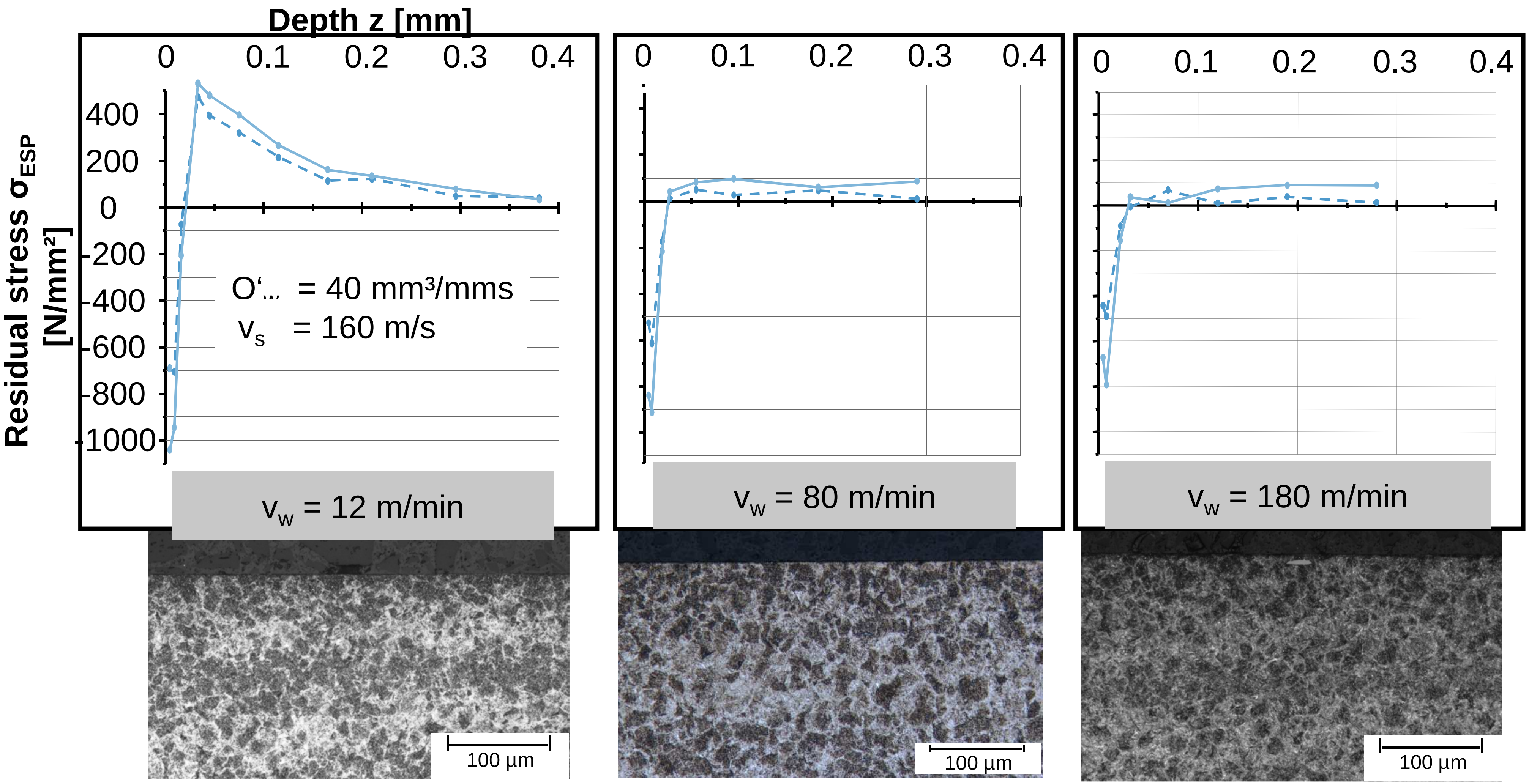




Fraunhofer IPT

WZL RWTH AACHEN

Results:
Validation of the FEM Simulation



Agenda

System „External-cylindrical Grinding“

Input-Parameters

- **Machine**
- **Workpiece**
- **Tool**
- **Preparation**
- **Coolant**
- **Process parameters**
- **Boundary condition**

?

What happens here…?

Fraunhofer IPT

WZL RWTHAACHEN

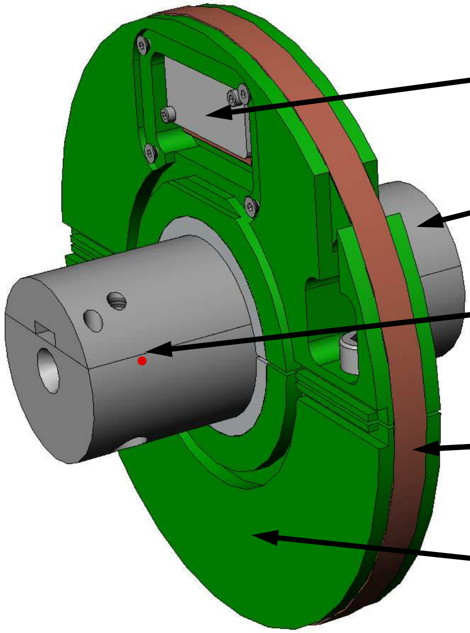
Grinding machine and parameters for external-cylindrical grinding

Machine
EMAG KOPP SN 204
Material
38MnS6 (BY)
Coolant
Emulsion (5%ig)
Needle nozzel

$d_{s,max}$
500 mm
P_{nenn}
30 kW
$n_{s,außen,max}$
7,500 min^{-1}
$v_{c,\ max}$
150 m/s
KSS
Emulsion/oil

Grinding parameters				**Dressing parameters**		
Q'_w [mm^3/mms]	v_w [m/min]	v_c [m/s]	a_e [mm/U]	U_d	a_{ed} [µm]	q_d
0.3 – 20	4.3 - 17.2	80	0.004 - 0.2	5	3	0.5
Grinding wheel				**Dressing tool**		
CBN 151 VSS 3443 J1 SN V 360 E (500 x 20 x 203.2)				301 SG 071P-140-0,5 rotating		

Temperature measurement for external-cylindrical grinding



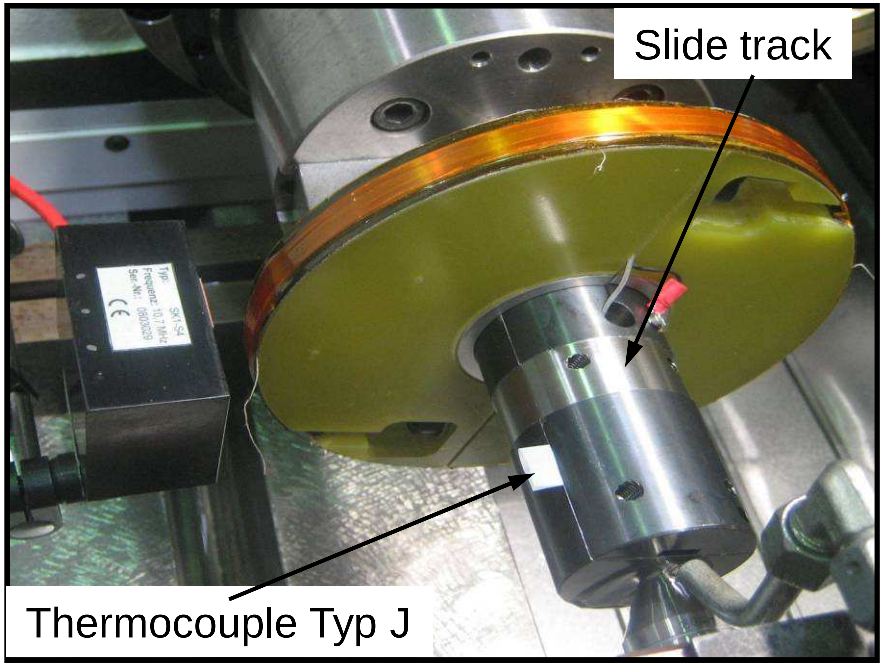
Rotorelectronics
K1-RK1-R2

Shaft

Constantan wire
(Typ J)

Coil for transmission of
measurement data

GreenGlass (GFK)-disc,
devided

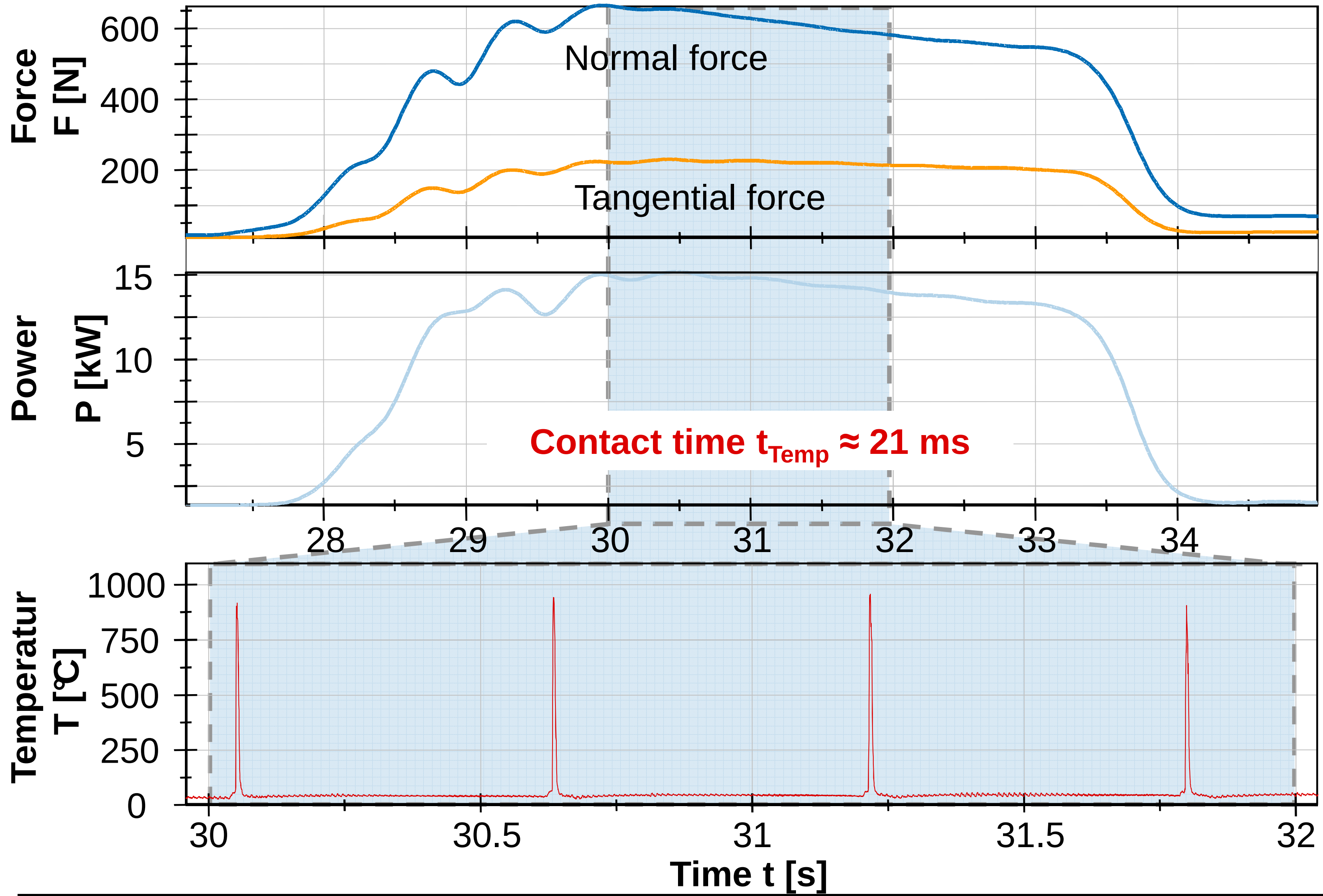


Stator unit
SK1-S4

Feedback unit
K1-WK1-T

Output

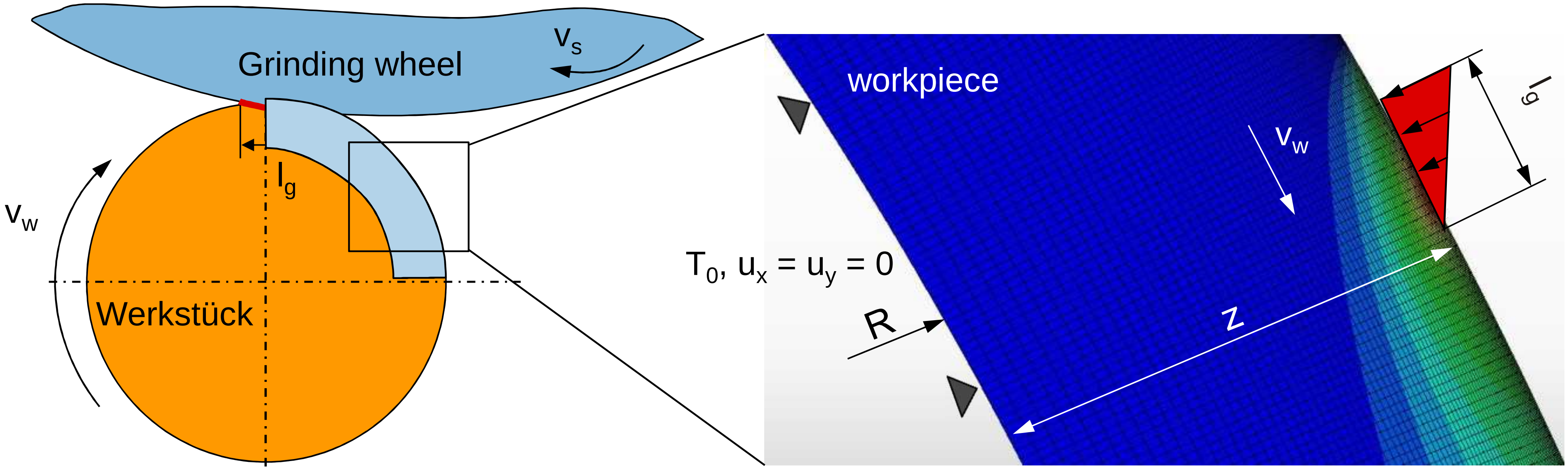
Results:
Experimental results of external-cylindrical grinding



Material
38MnS6 (BY)
Grinding parameters
Q'_w = 20 mm³/mms
v_w = 0.168 m/s
v_s = 80 m/s
CBN - grinding wheel
Coolant
Emulsion (5%ig)
Needle nozzle
Dressing parameters
U_d = 5
q_d = 0.5
a_{ed} = 3 µm

- Normal force
- Tangential force
- Grinding spindel power
- Temperature

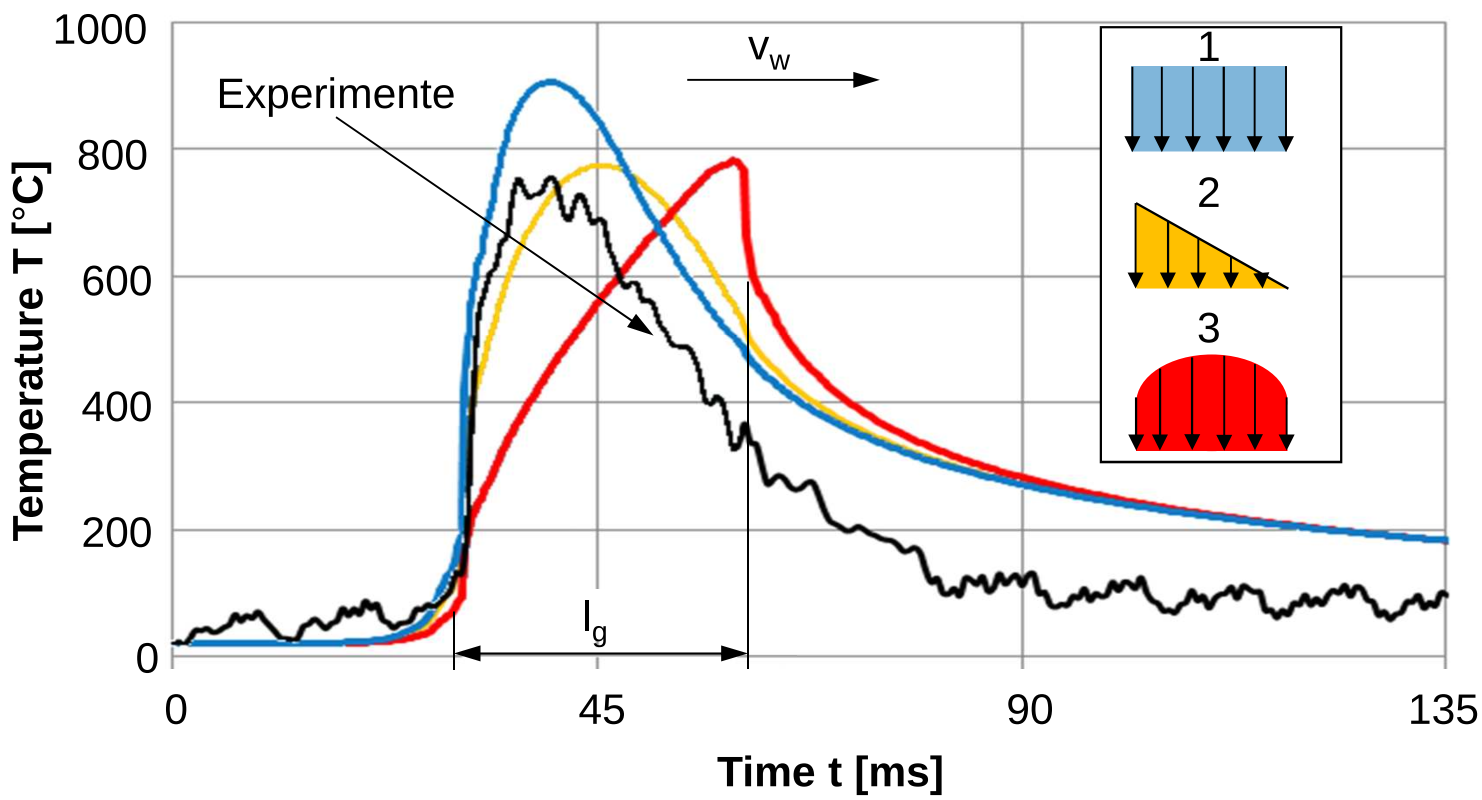
From the real process to a Finite Element Model



Boundaries

- Two-dimensional model with a quadratic moving heat source
- Temperature-independent thermal material properties
- The surface of the solid considered with heat convection
- Bottom surface is set to 20℃
- Maximum temperature of the coolant lubricant is t_B = 120℃ due to the boiling point of emulsion

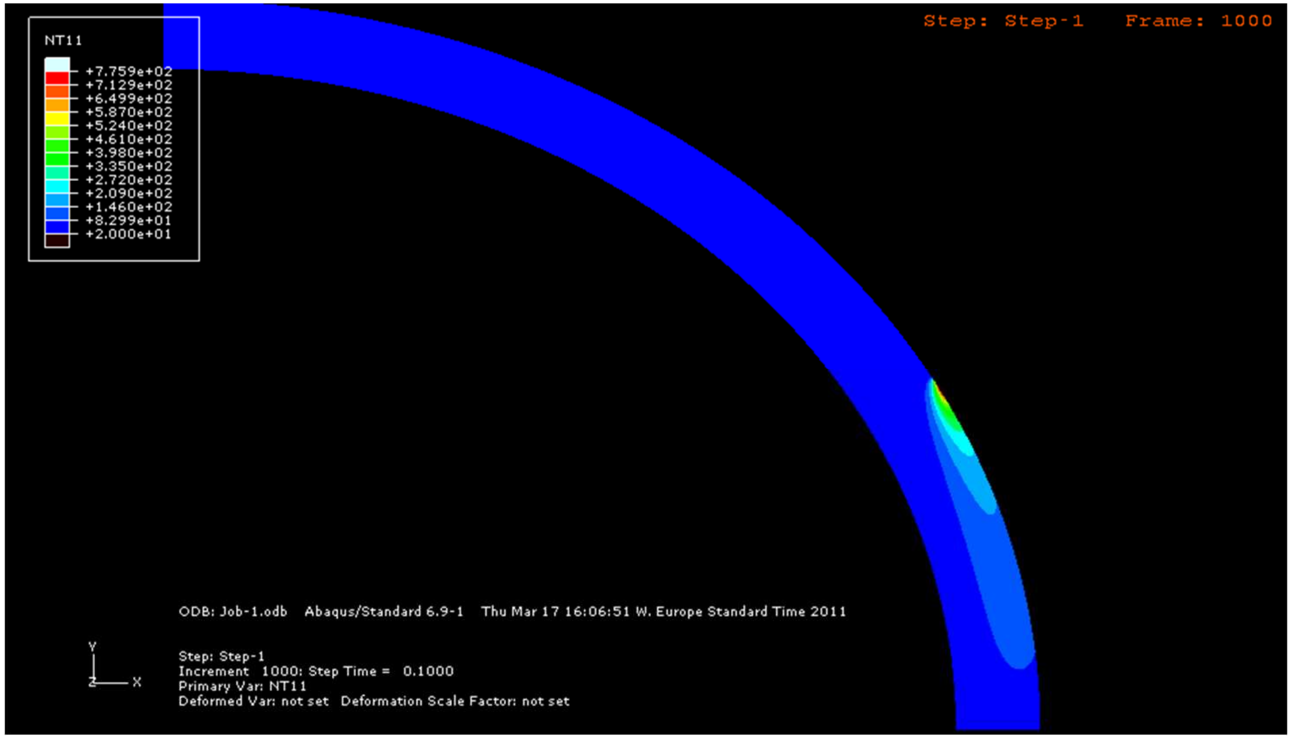
Results:
Validation of cylindrical grinding simulation results



Material
38MnS6 (BY)
Grinding parameters
Q'_w = 10 mm³/mms
v_w = 0.168 m/s
v_s = 80 m/s
CBN - grinding wheel
Coolant
Emulsion (5%ig)
Needle nozzle
Dressing parameters
U_d = 5
q_d = 0.5
a_{ed} = 3 µm

The triangular heat source fits best.

Results:
Cylindrical grinding simulation results



20 °C
82
146
209
272
335
398
461
524
587
649
712
775

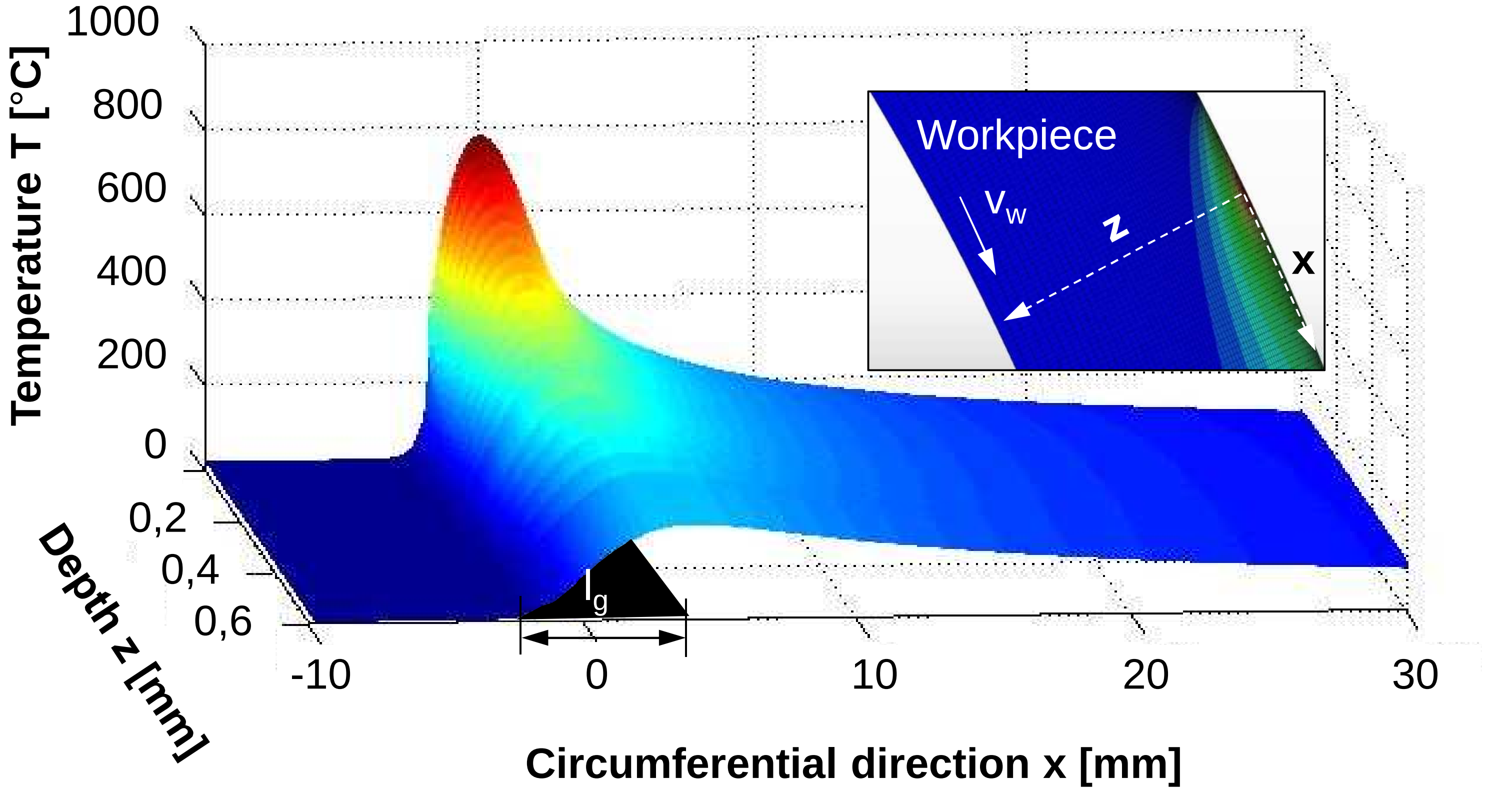
Material
38MnS6 (BY)
Grinding parameters
Q'_w = 10 mm³/mms
v_w = 0.168 m/s
v_s = 80 m/s
CBN - grinding wheel
Coolant
Emulsion (5%ig)
Needle nozzle
Dressing parameters
U_d = 5
q_d = 0.5
a_{ed} = 3 µm

The visual representation of the thermal effects allows a better understanding.

Fraunhofer IPT

WZL RWTHAACHEN

Results:
Cylindrical grinding simulation results



Material
38MnS6 (BY)
Grinding parameters
Q'_w = 10 mm³/mms
v_w = 0.168 m/s
v_s = 80 m/s
CBN - grinding wheel
Coolant
Emulsion (5%ig)
Needle nozzle
Dressing parameters
U_d = 5
q_d = 0.5
a_{ed} = 3 µm

The temperatures in the workpiece can be determined at any time, anywhere. Thus, the temperature history for the workpiece is known.

Open discussion

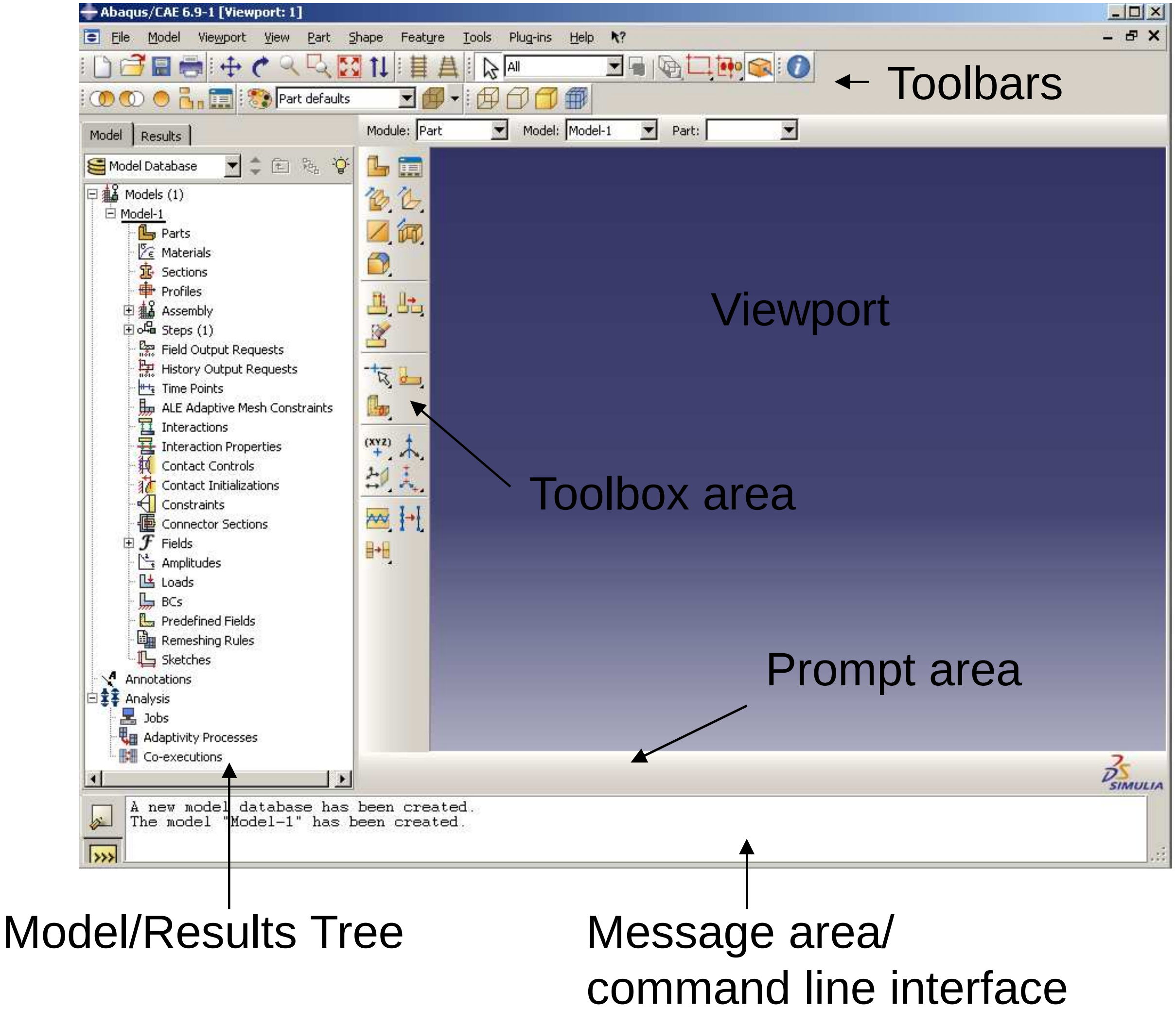
Thanks for your attention!

Modelling and simulation of grinding processes are complex but not impossible.

Dipl.-Ing. Michael Duscha
Email: m.duscha@wzl.rwth-aachen.de
Tel.: +49 241-80-28185

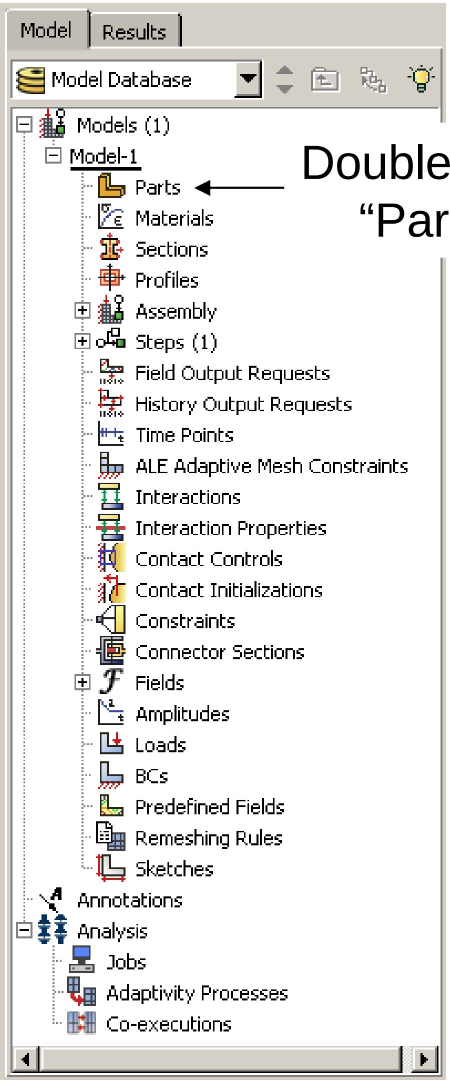
Agenda

Starting a new model: main window

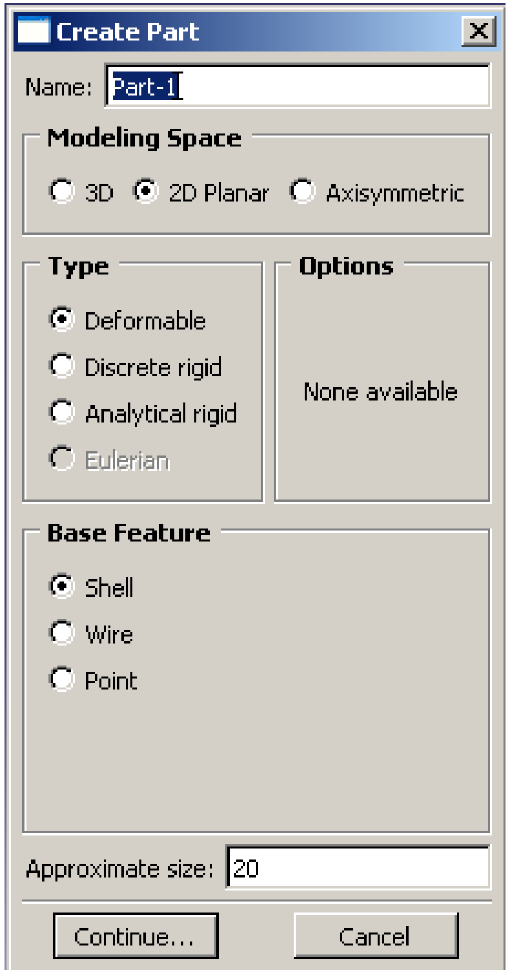


- **Viewport:** where the model and results will be displayed
- **Model Tree/Results Tree:** graphical overview of model/results
- **Prompt area:** shows prompts related to the current tool being used
- **Message area:** displays status information and warnings
- **Command line interface:** allows use of command line inputs
- **Toolbox area:** displays tools available in the current module

Creating a part

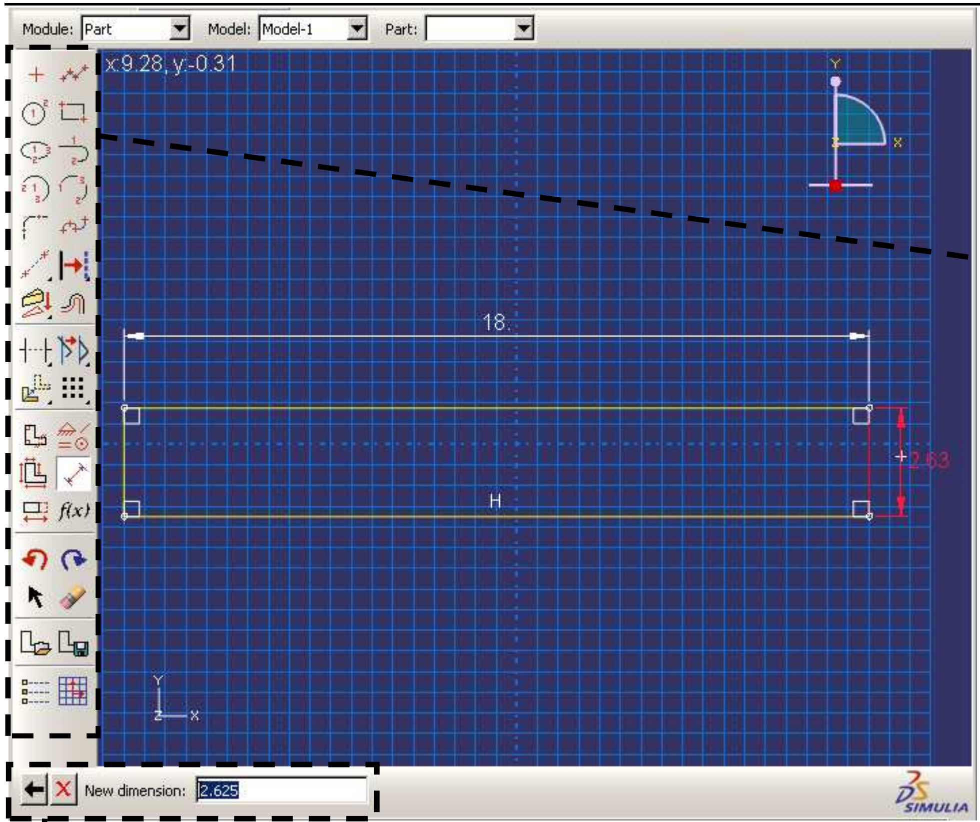


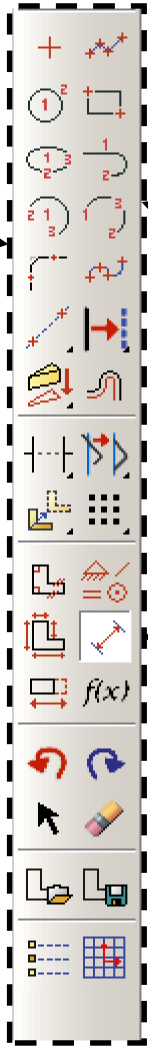
Double-click “Parts”



- **“Base Feature”** controls the feature type used to sketch the basic form of the part.
- **“Approximate size”** controls the size and spacing of the grid used for sketching the part. It should be approximately equal to the **largest dimension** of the part in the model units.

Sketching the part





The toolbox contains tools for sketching the part

Tools for creating basic shapes

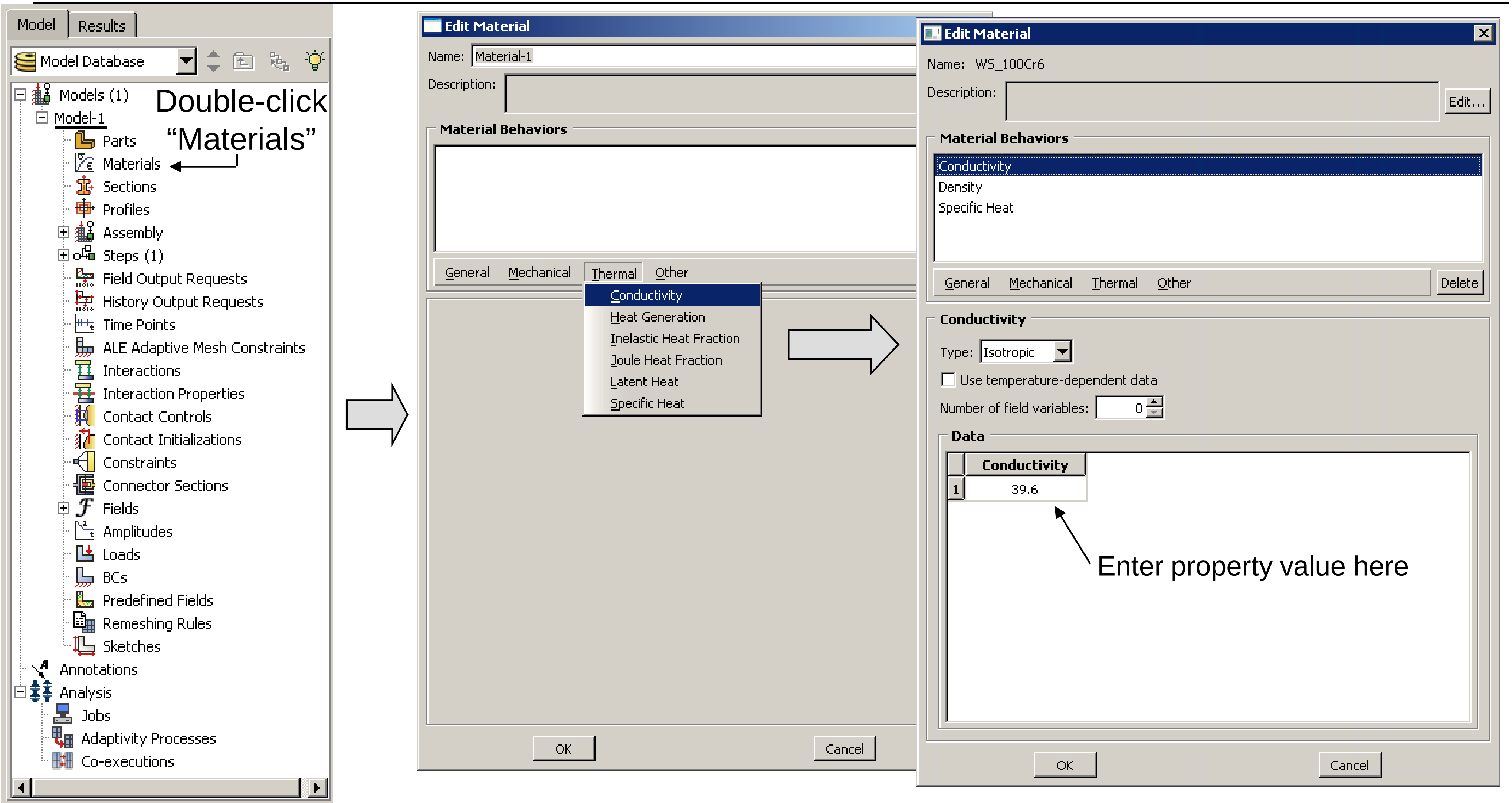
Tools for adding constraints

Cancel procedure

Go to previous step

Example use of the prompt area to define a dimension

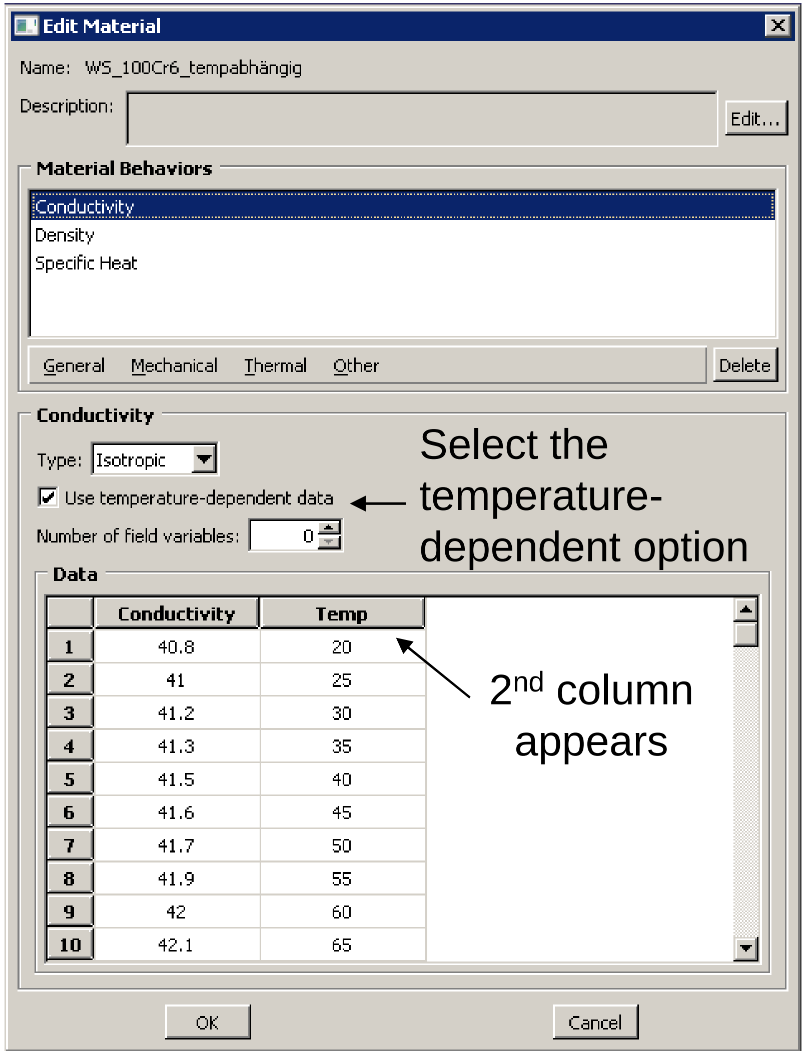
Defining material properties



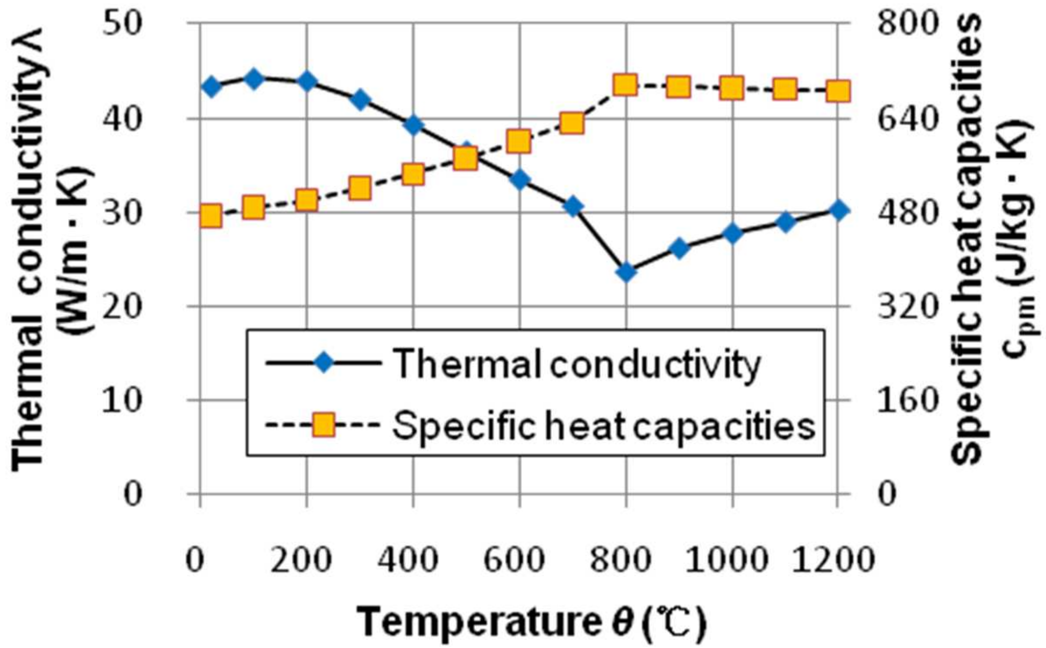
Fraunhofer IPT

WZL RWTHAACHEN

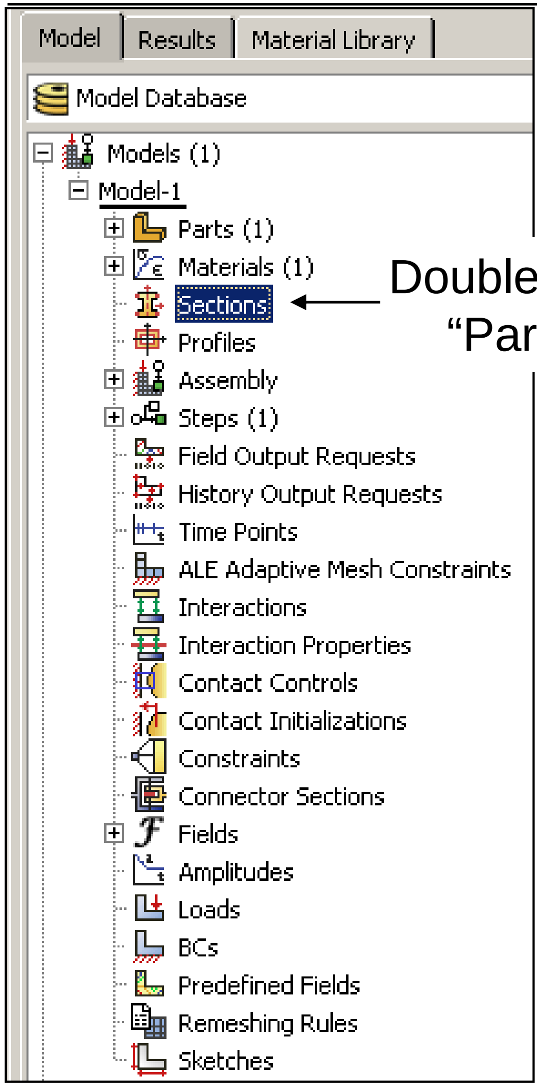
Defining temperature-dependent material properties



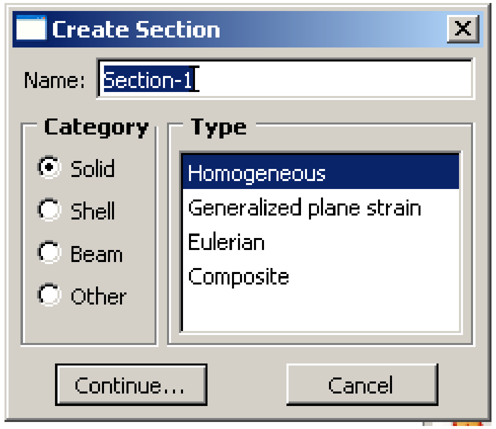
- The individual entries in the data table do not need to be entered individually. A table can be copied from Microsoft Excel, for instance, and pasted into the Material Editor.



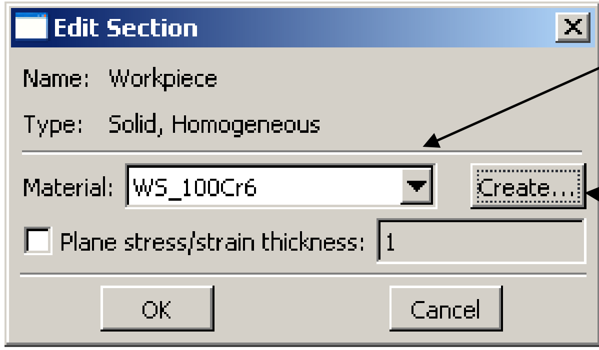
Defining a section



Double-click “Parts”



This model represents a homogeneous solid, so the section should be “Homogeneous Solid” even though the model is 2-D (“Shell” is for parts with a thickness that is much smaller than the other two dimensions).

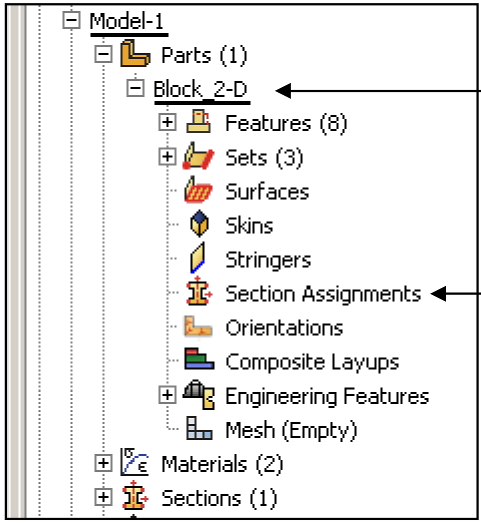


Select an existing material

or

create a new one

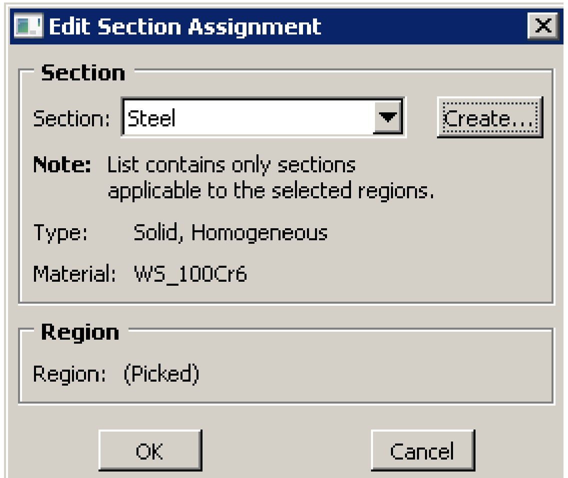
Assigning a section to a part



Under “Parts”, expand the Model Tree for the part

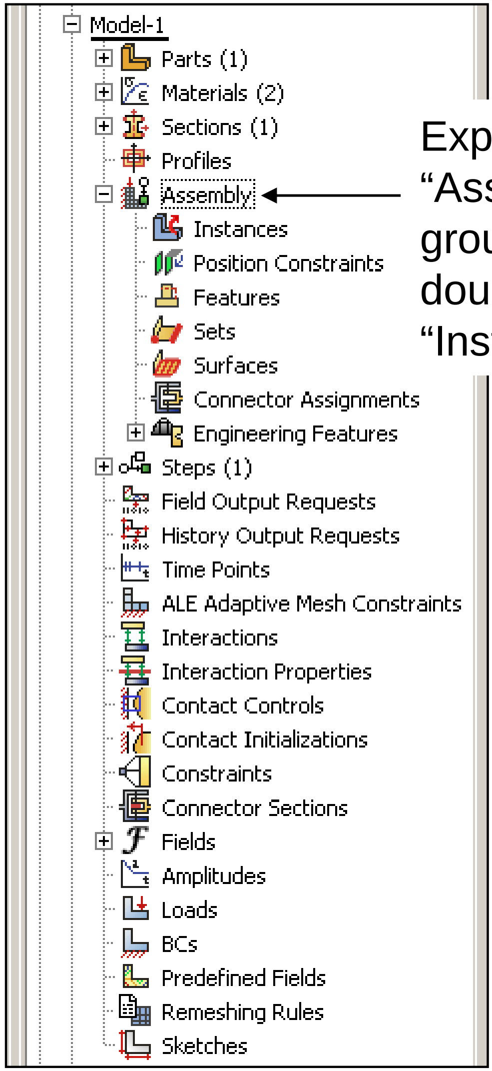
Double-click “Section Assignments”

Follow the directions in the Prompt Box and select the regions of the part using the mouse (selected regions will be highlighted in red)



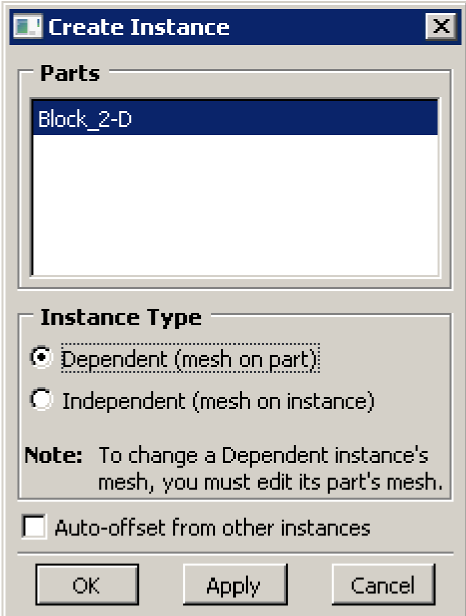
- Assigning the section to a part region defines what material properties are used for that part region.
- The material properties assigned to a region can be easily changed by changing the *Section* definition. The Section Assignment does not need to be changed.

Creating an assembly



Expand the "Assembly" group and double-click "Instances"

- The assembly contains all the parts involved in an analysis and defines their relative locations and orientations.
  - contains one or more parts
  - may contain multiple copies ("part instances") of a single part
  - the orientation of a part instance in the assembly is not necessarily the same as its orientation in the "Part" module



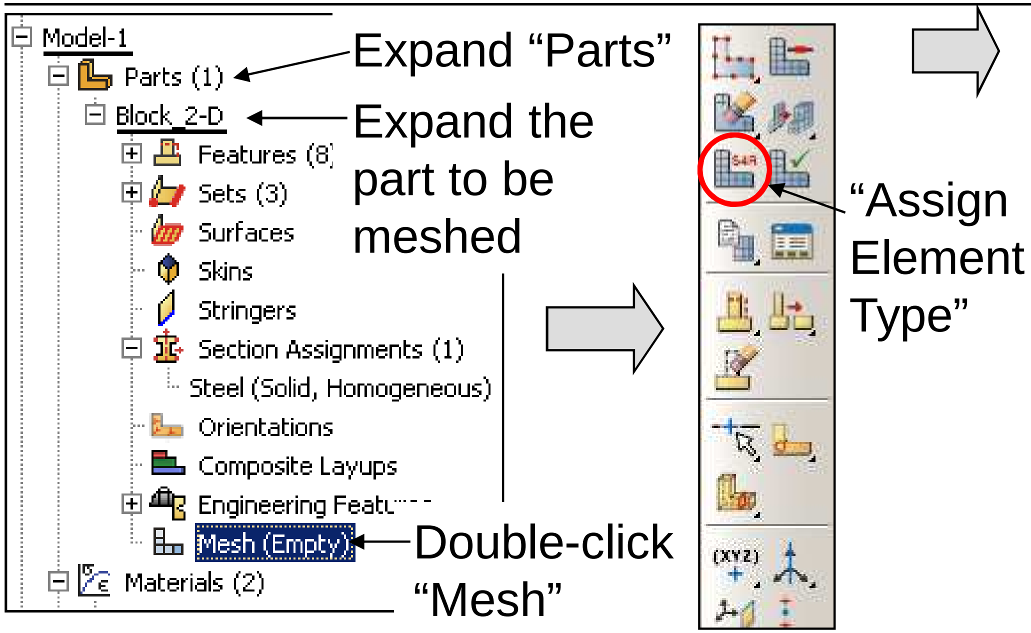
Additional copies of a part or other parts can also be added to the assembly by repeating the process.

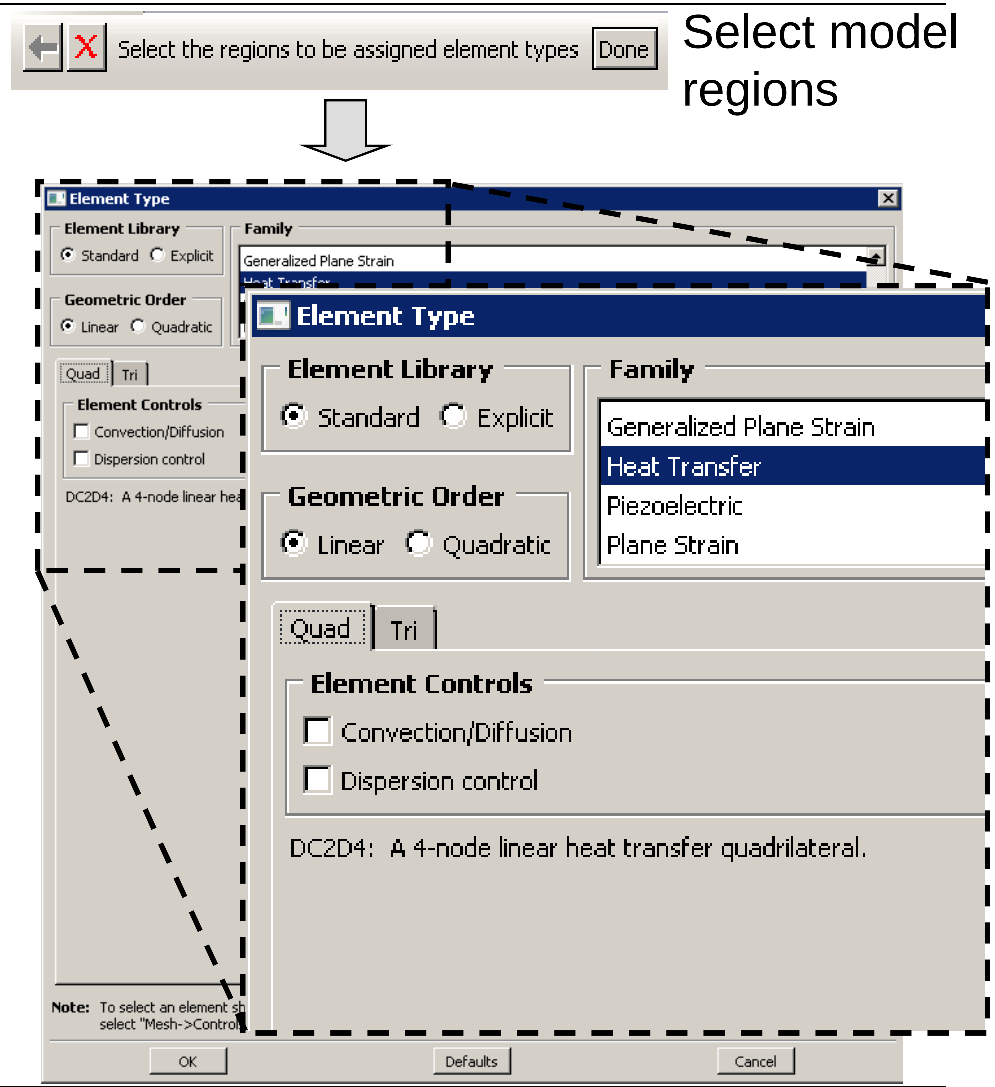
The current grinding simulation includes only a single part instance of the workpiece.

Fraunhofer IPT

WZL RWTHAACHEN

Meshing the model: element type



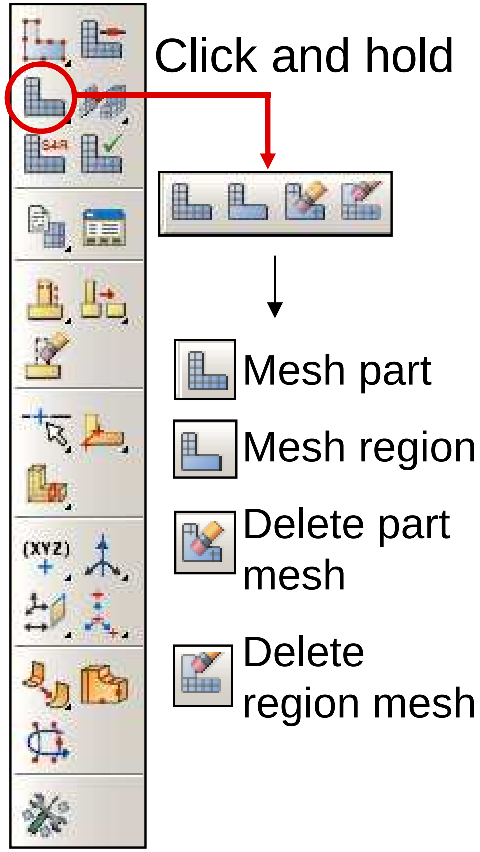


- Element type can be defined before or after defining the mesh.
- Abaqus Explicit supports fewer element types than Abaqus Standard (Heat Transfer elements are only available in Standard)
- The current model uses linear quadrilateral Heat Transfer elements
- **Documentation on Element Types:** Abaqus User's Manual, sections 23-28.

Meshing the model: seeding the part

- Seeding the part guides Abaqus in generating the mesh. The “seeds” are placed along the edges of the part or part regions, and Abaqus will then place the element nodes at the seeds whenever possible.
- Methods of seeding
  - “Seed Part”—seeds all edges in the part based on the desired average element size (creates “Global seeds”)
  - “Seed Edge: by number”—seeds selected edges, based on the desired number of elements along that edge (“Local seeds”)
  - “Seed Edge: By Size”—seeds selected edges, based on the desired average element size along that edge (“Local seeds”)
  - “Seed Edge: Biased”—creates a non-uniform seed distribution along selected edges (“Local seeds”); the User defines the “Bias Ratio” (the desired ratio between the largest and the smallest element lengths) and the number of elements to be put along the edge
- **Documentation on seeding:** Abaqus CAE User’s Manual, section 17.4 “Understanding seeding”

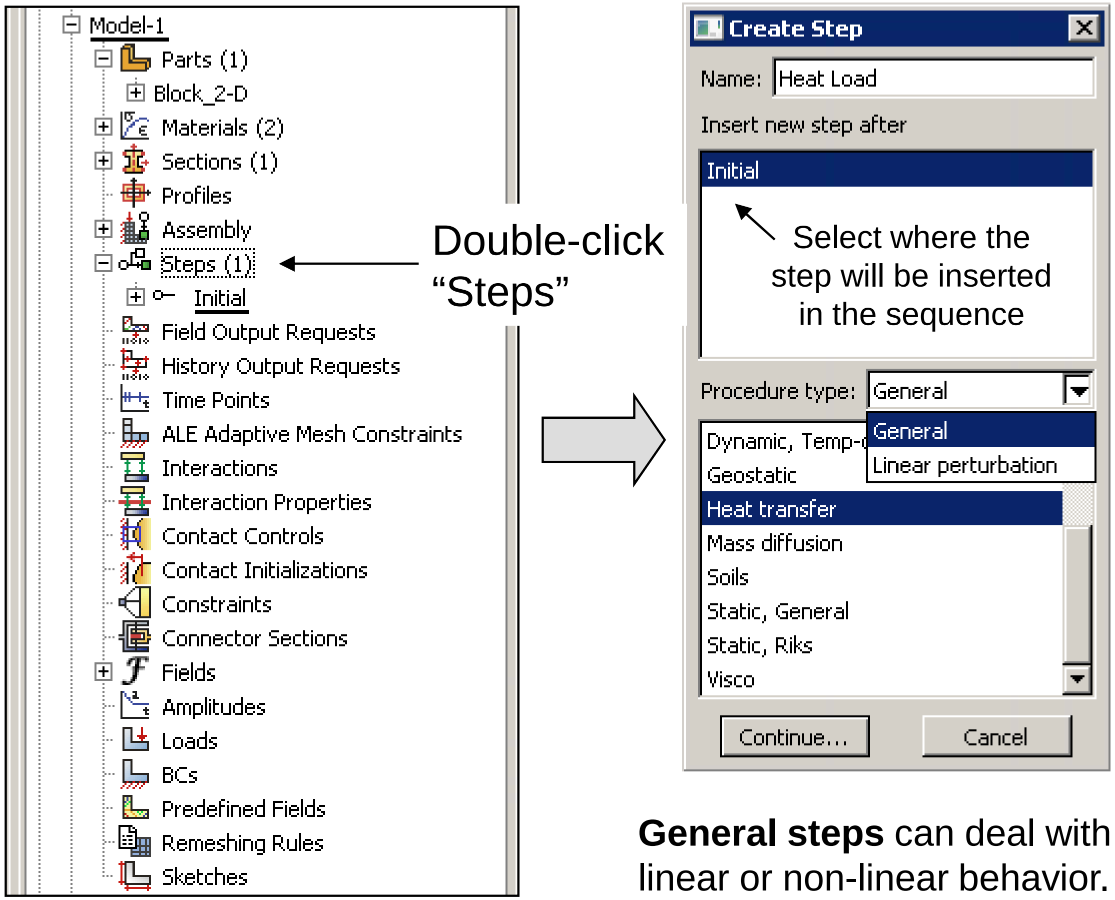
Meshing the model: generating the mesh



- Meshing techniques:
  - Structured meshing—simple, predefined mesh geometries are adapted to the geometries of the part (prefer)
  - Swept meshing—a mesh is generated on one side of the region and copied one element layer at a time along the “sweep path” until it reaches the target side
  - Free meshing—unpredictable, unstructured meshing technique that uses no predefined mesh geometries
  - Bottom-up meshing—a manual, incremental meshing technique, in which the tie between the mesh and the part geometry is not as strict as in the automatic meshing techniques.
- Abaqus indicates possible meshing techniques for part regions using color coding, and provides more detailed documentation about these techniques in the Abaqus/CAE User’s Manual, chapter 17. Adding partitions or changing the element type can affect the available meshing techniques.
  - Structured meshing—color code: green, section 17.8.1 Swept meshing—color code: yellow, section 17.9.1
  - Free meshing—color code: pink, section 17.10.1
  - Bottom-up meshing—color code: light tan, section 17.11.1
  - Unmeshable—color code: orange

Creating analysis steps

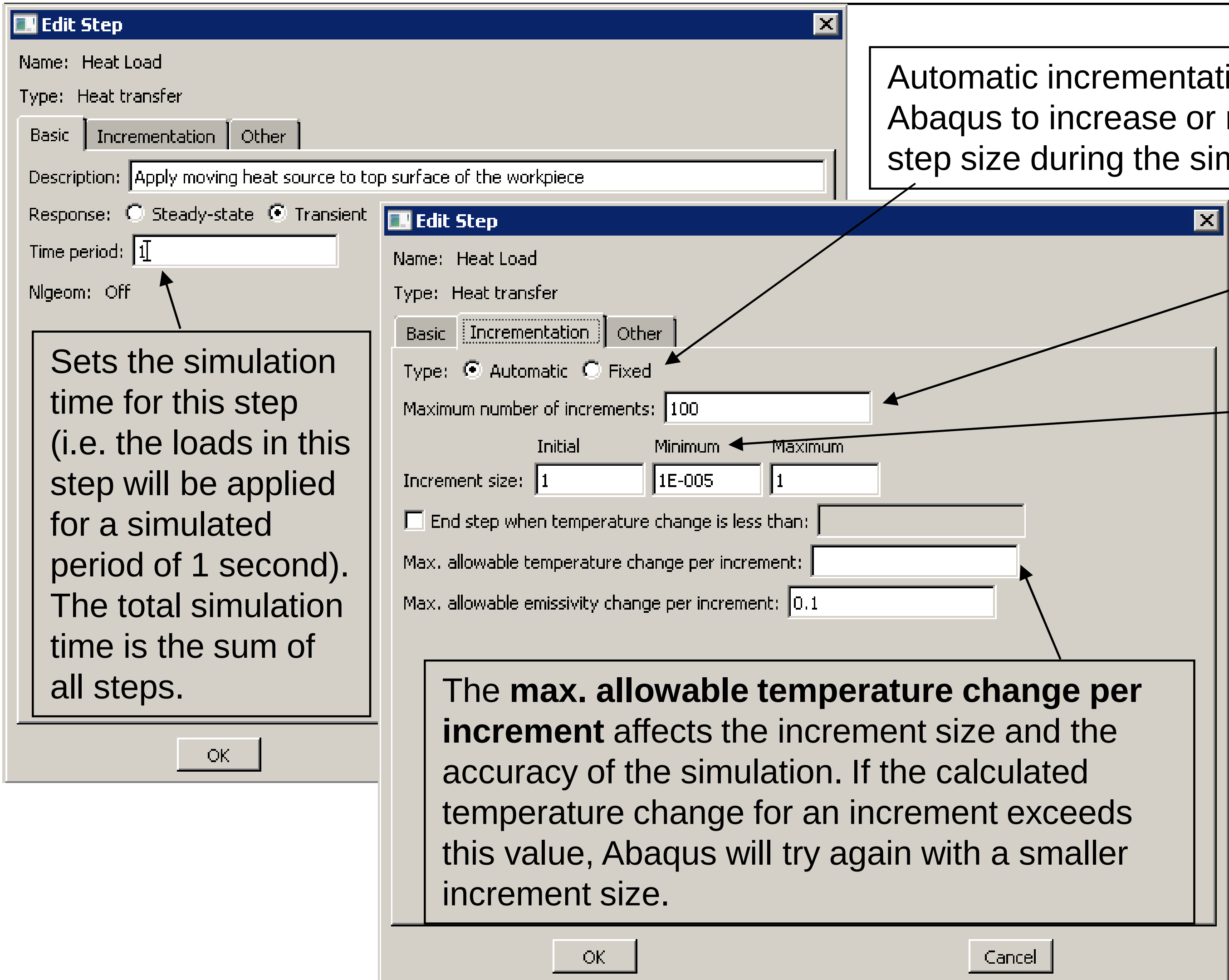
- Abaqus analyses involve multiple steps in which different loads and constraints are applied to the model
- Abaqus automatically creates the step “Initial”. This step can be used to apply initial conditions or boundary conditions with the limitation that all constraints applied in “Initial” must have a value of 0.
  - I.e. Step “Initial” can define the initial temperature to be 0 or constrain a set of points to have zero displacement. It *cannot* define the initial temperature to be 20 or define a non-zero starting velocity for a point.
- Analysis steps and output requests must be created by the user



General steps can deal with linear or non-linear behavior.

Linear perturbation steps are only for linear behavior.

Creating analysis steps



Sets the simulation time for this step (i.e. the loads in this step will be applied for a simulated period of 1 second). The total simulation time is the sum of all steps.

Automatic incrementation allows Abaqus to increase or reduce the step size during the simulation

The **maximum number of increments** should not be set too low, or the simulation will be terminated partway through.

Likewise, the **minimum increment size** will also terminate the simulation if set too high.

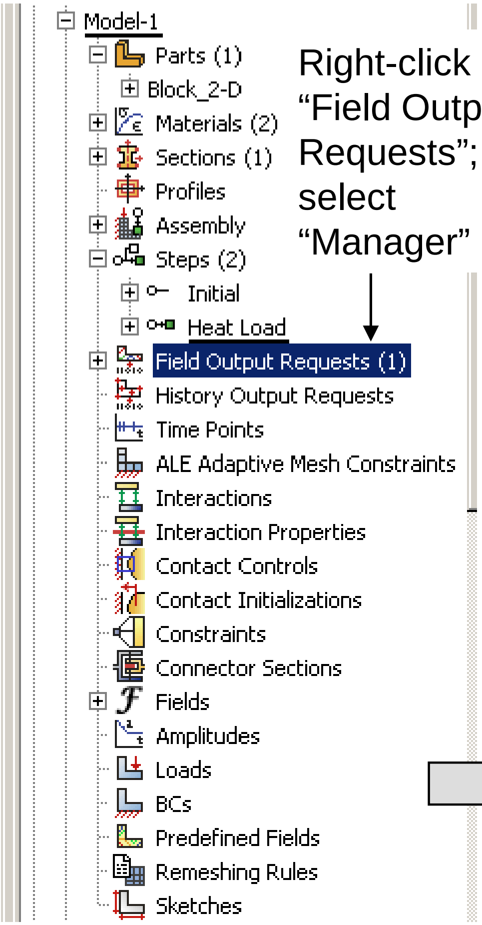
The current model uses a maximum number of increments of 80000 for 3 seconds of simulation time and a minimum increment size of 3E-008.

The **max. allowable temperature change per increment** affects the increment size and the accuracy of the simulation. If the calculated temperature change for an increment exceeds this value, Abaqus will try again with a smaller increment size.

Fraunhofer IPT

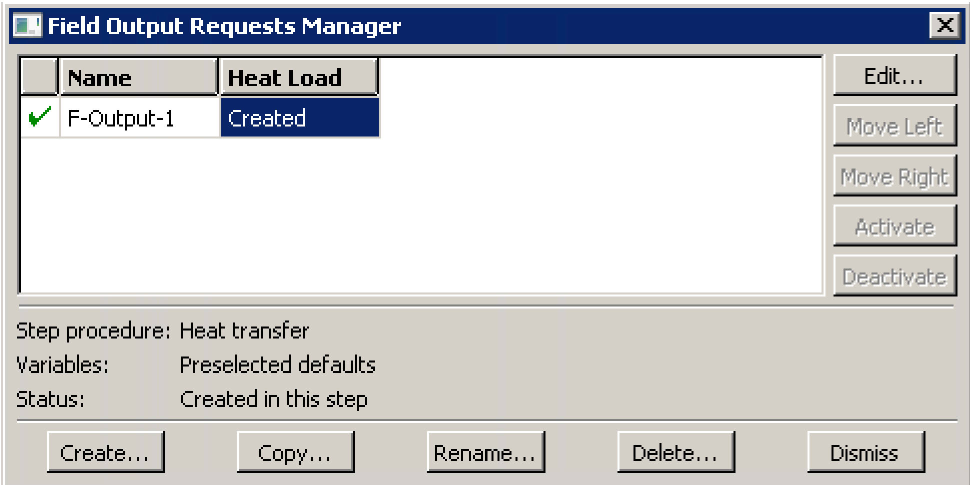
WZL RWTHAACHEN

Controlling output data



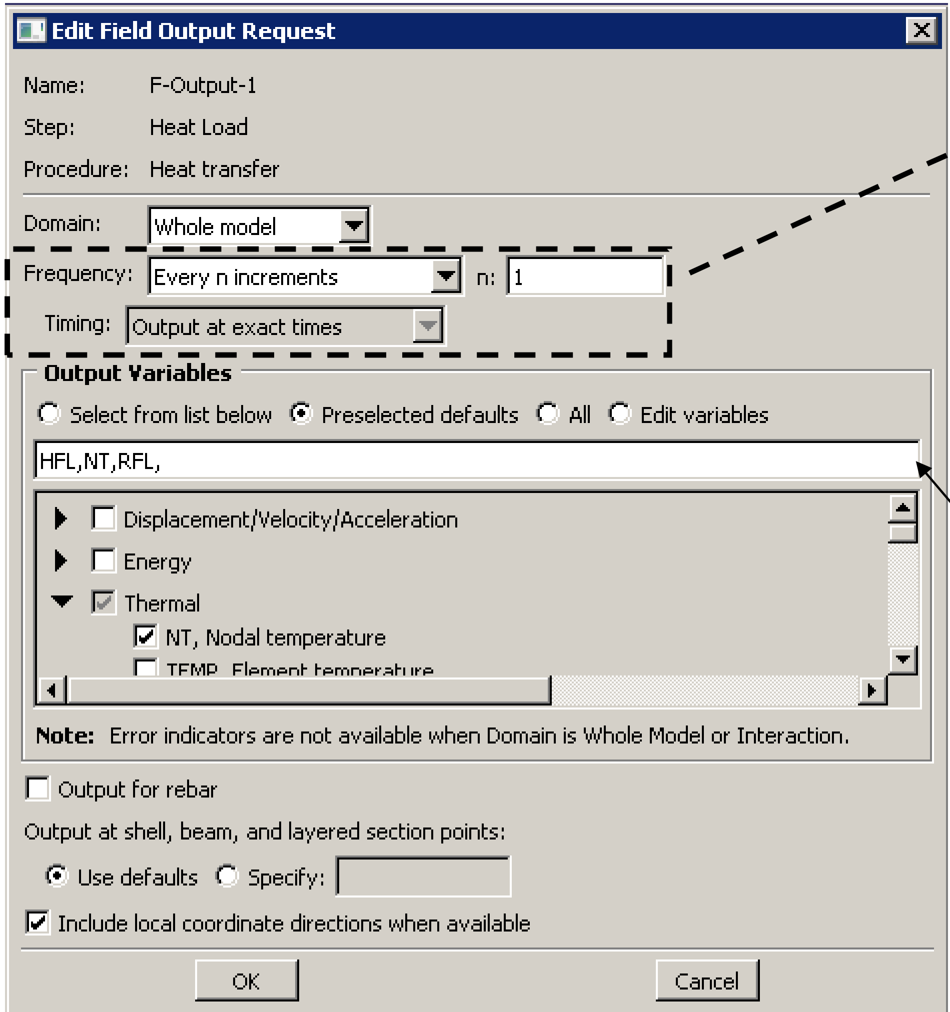
Right-click "Field Output Requests"; select "Manager"

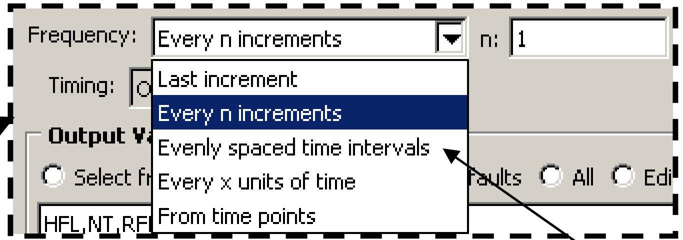
- There are two types of output request
  - Field output: records data from the entire model or from large portions of the model (intended to be at relatively low frequency)
  - History output: records data for a smaller region of the model at high frequency
- When an analysis step is created, Abaqus automatically creates a default "Field Output Request" that records default output values for that step type. This output request can also be edited to add or remove output requests



Click "Edit"

Field Output Request options



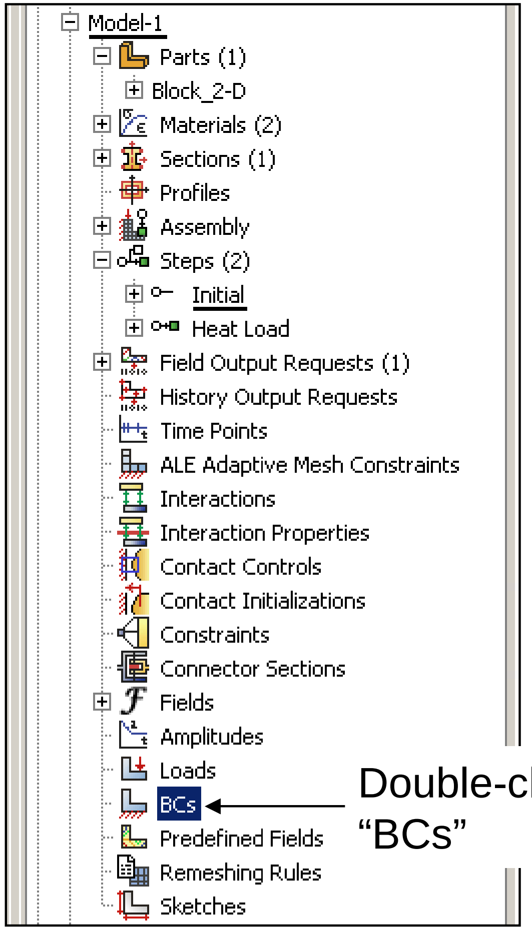


The evenly spaced time intervals option could help reduce the size of the output file and make the data from multiple simulations easier to compare (I had not seen this option, so I have not tried it).

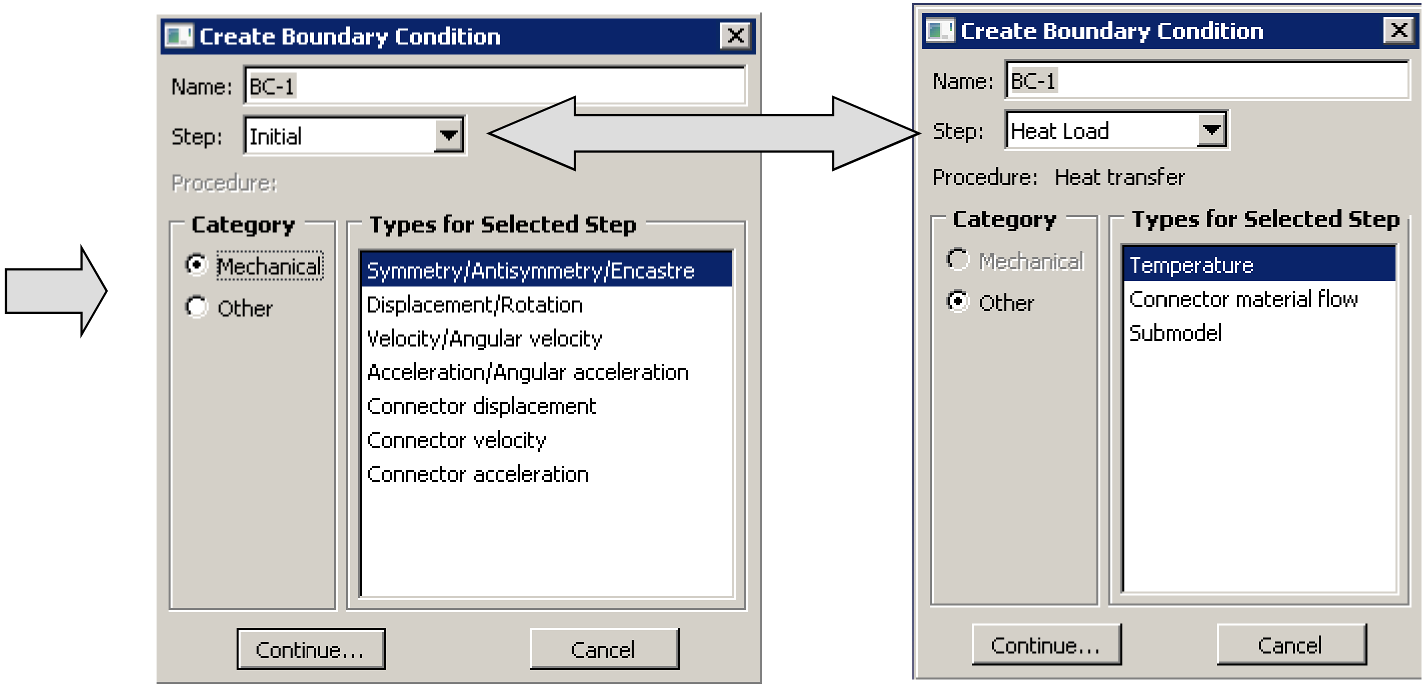
The abbreviations of all currently selected variables are shown in this field

So far, the default heat transfer outputs (nodal temperature, heat flux vector and reaction fluxes) have been used for the simulations

Creating a boundary condition



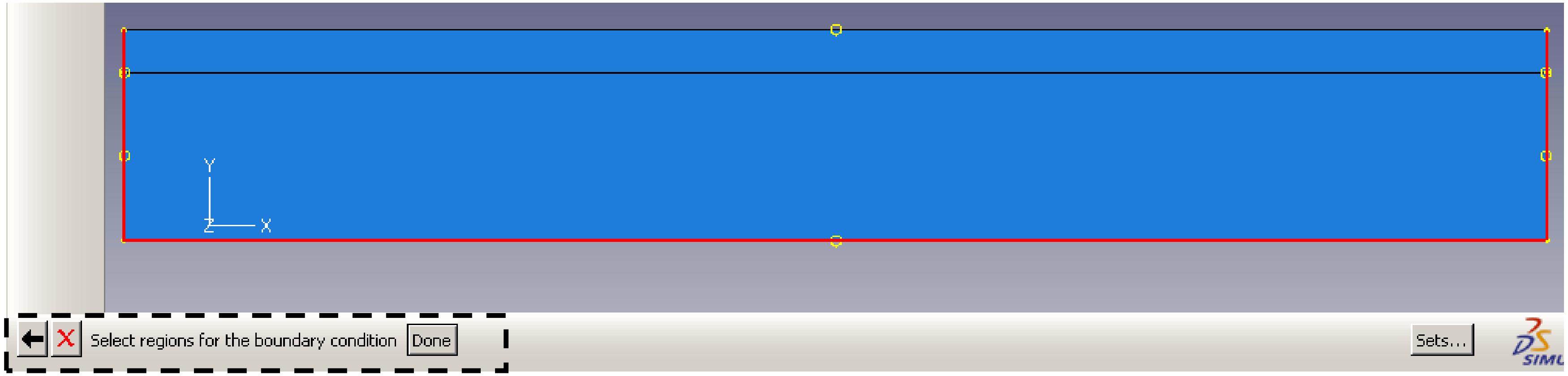
Double-click "BCs"



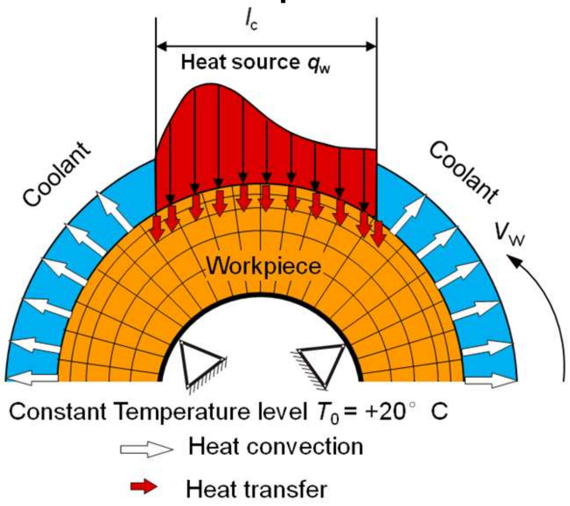
The available BC types are limited based on the step type. E.g. No mechanical BCs can be set in a heat transfer step.

Documentation: Abaqus/CAE User's Manual, section 16.8.2 "Creating boundary conditions"; 16.10 "Using the boundary condition editors"

Selecting the region to apply a boundary condition

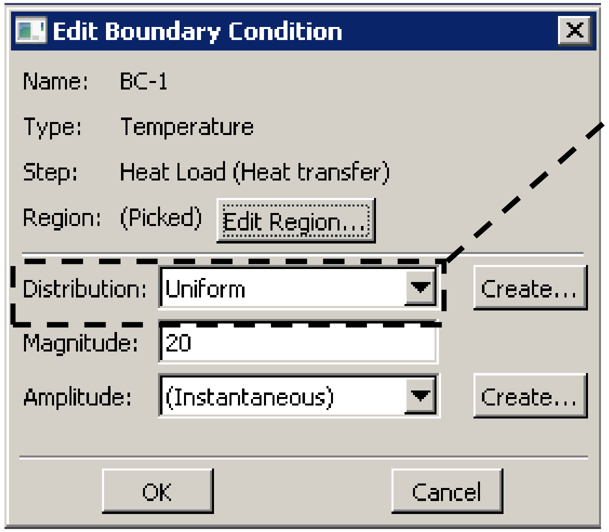


Instructions appear in the Prompt Box.



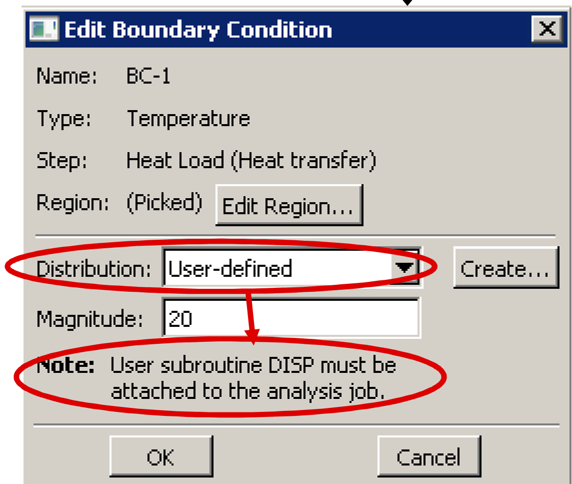
- Regions are selected by clicking on features (edges or areas) of the model.
- Features are highlighted in orange as the cursor moves over them; a selected feature is highlighted red.
- Hold down the shift key to select multiple features.

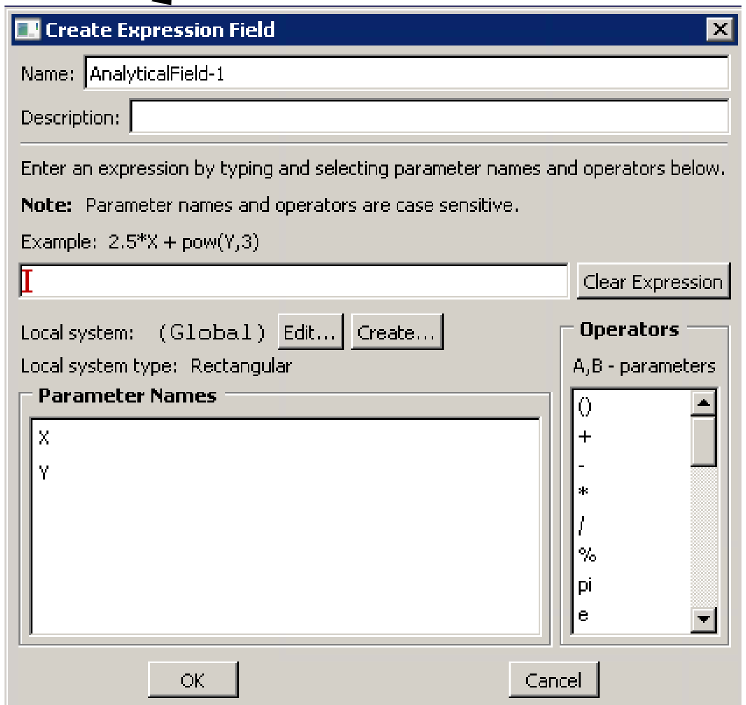
Setting the distribution of a boundary condition



Defining the distribution as "User-defined" would require attaching a subroutine.

A (non-time-dependent) non-uniform distribution can also be defined here.





Boundary conditions: Degrees of freedom

- The degrees of freedom are numbered as shown in the figure. The 1-, 2- and 3-directions are the x-, y- and z-directions, respectively.

Numbering for degrees of freedom (from Abaqus documentation)

1 Translation in the 1-direction (U1).

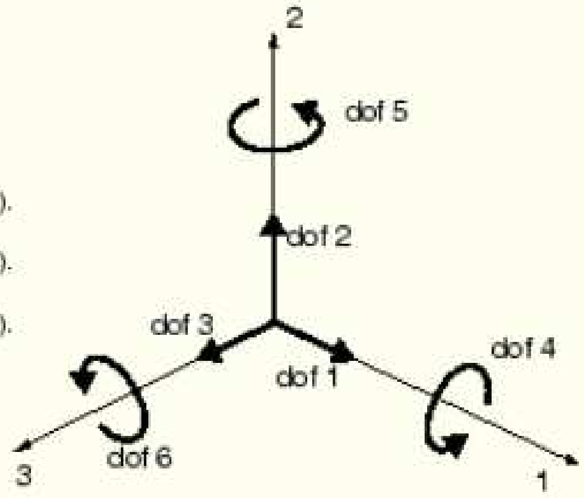
2 Translation in the 2-direction (U2).

3 Translation in the 3-direction (U3).

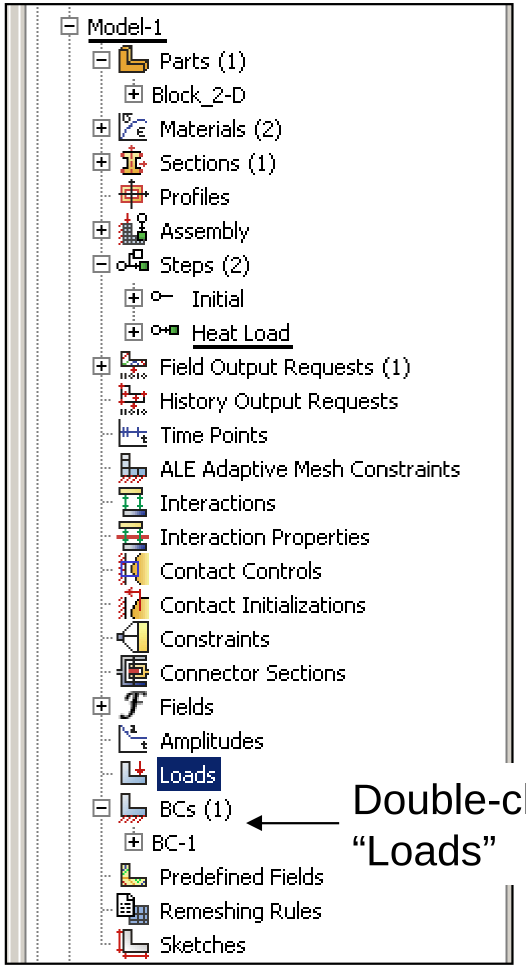
4 Rotation about the 1-direction (UR1).

5 Rotation about the 2-direction (UR2).

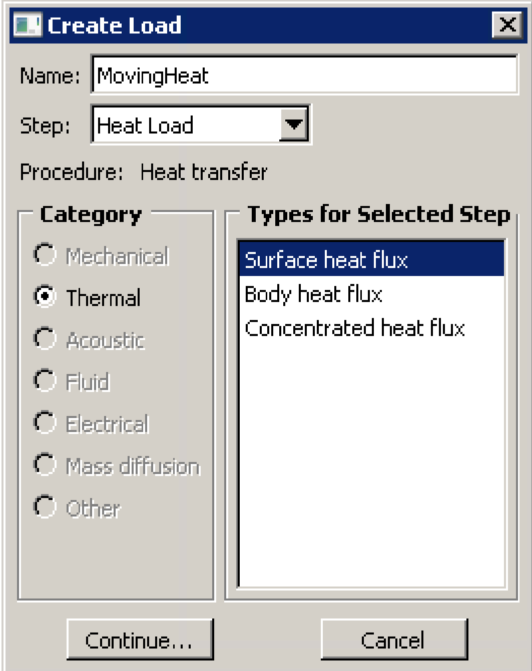
6 Rotation about the 3-direction (UR3).



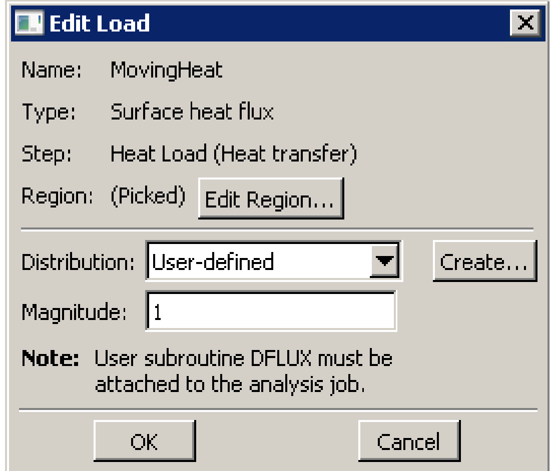
Creating a load



Double-click "Loads"



The types of loads are limited based on the step type.



The distribution options for loads are similar to those for boundary conditions.

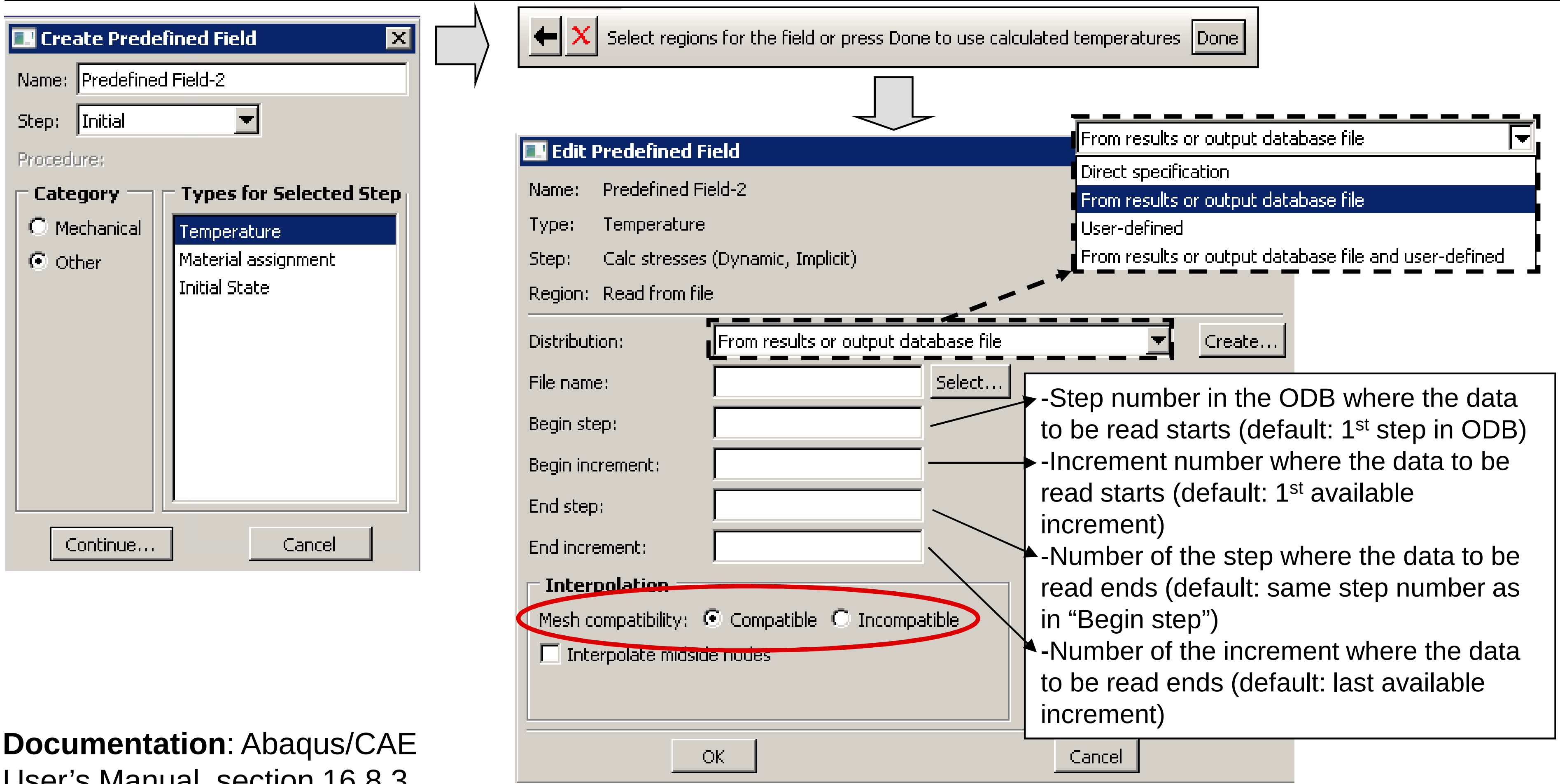
Documentation: Abaqus/CAE User's Manual, section 19.8.1 "Creating loads"; 16.9 "Using the load editors"

Subroutines

- Subroutines provide more flexibility for specifying certain model parameters than is provided by normal input methods.
- Different subroutines with different specifications are required for different purposes. I.e. a different subroutine is required for specifying a heat flux distribution (DFLUX) than is required for specifying a distribution for a boundary condition (DISP).
- Subroutines are written in Fortran, and a suitable Fortran compiler is required to use them in a simulation.
- Documentation:
  - Abaqus Analysis User's Manual, 14.2 "User subroutines and utilities"
  - Abaqus User Subroutines Reference Manual (includes discussion of individual subroutines and their requirements)

???

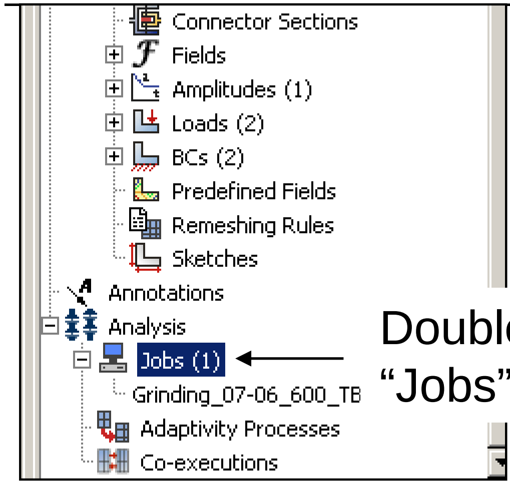
Incorporating previous results as a predefined field



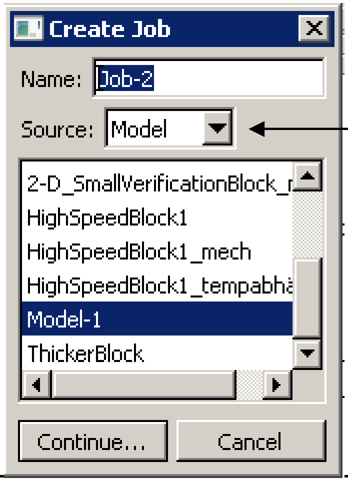
Documentation: Abaqus/CAE User’s Manual, section 16.8.3 “Creating predefined fields”; 16.11 “Using the predefined field editors”

Compatible: the mesh in the source ODB and the current model are the same or differ only in the element order
Incompatible: dissimilar meshes

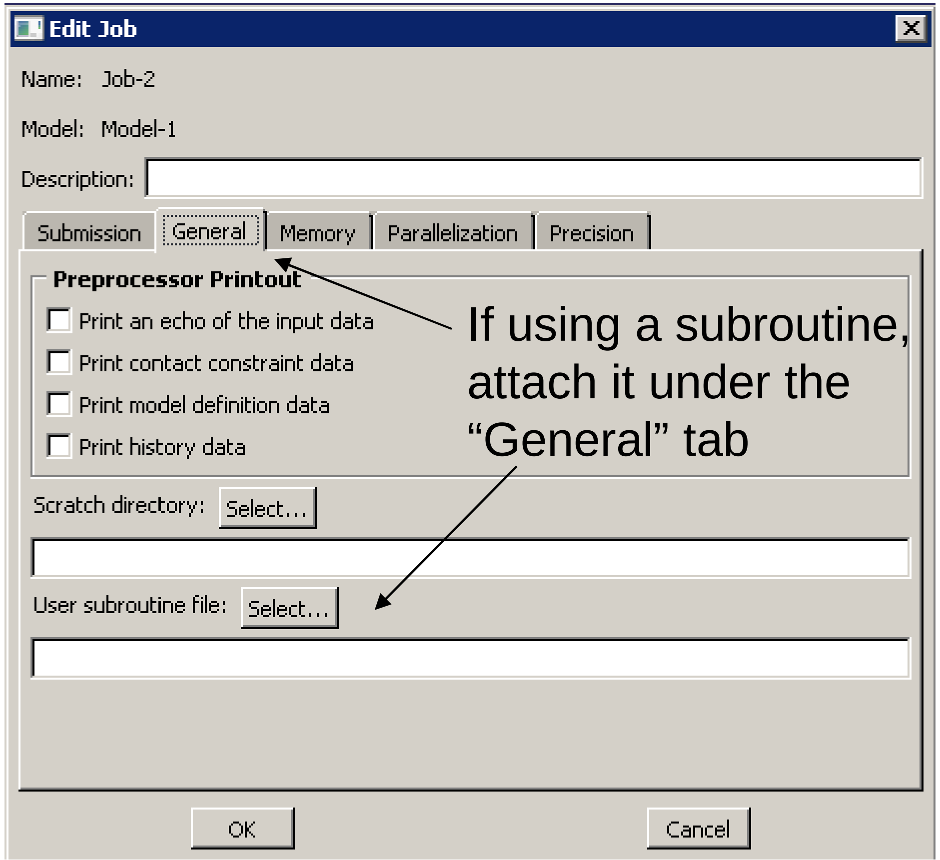
Creating a job



Double-click "Jobs"



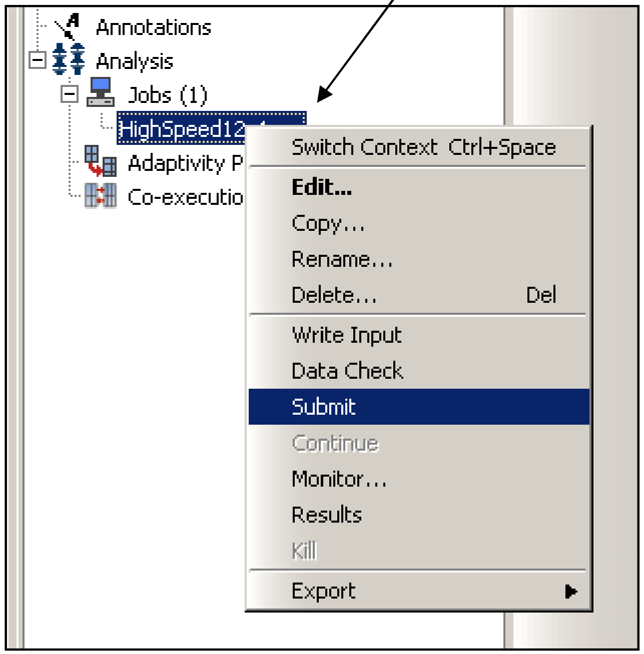
The source can be either a model or an input file.



If using a subroutine, attach it under the "General" tab

Submitting the job

Right-click on the job name

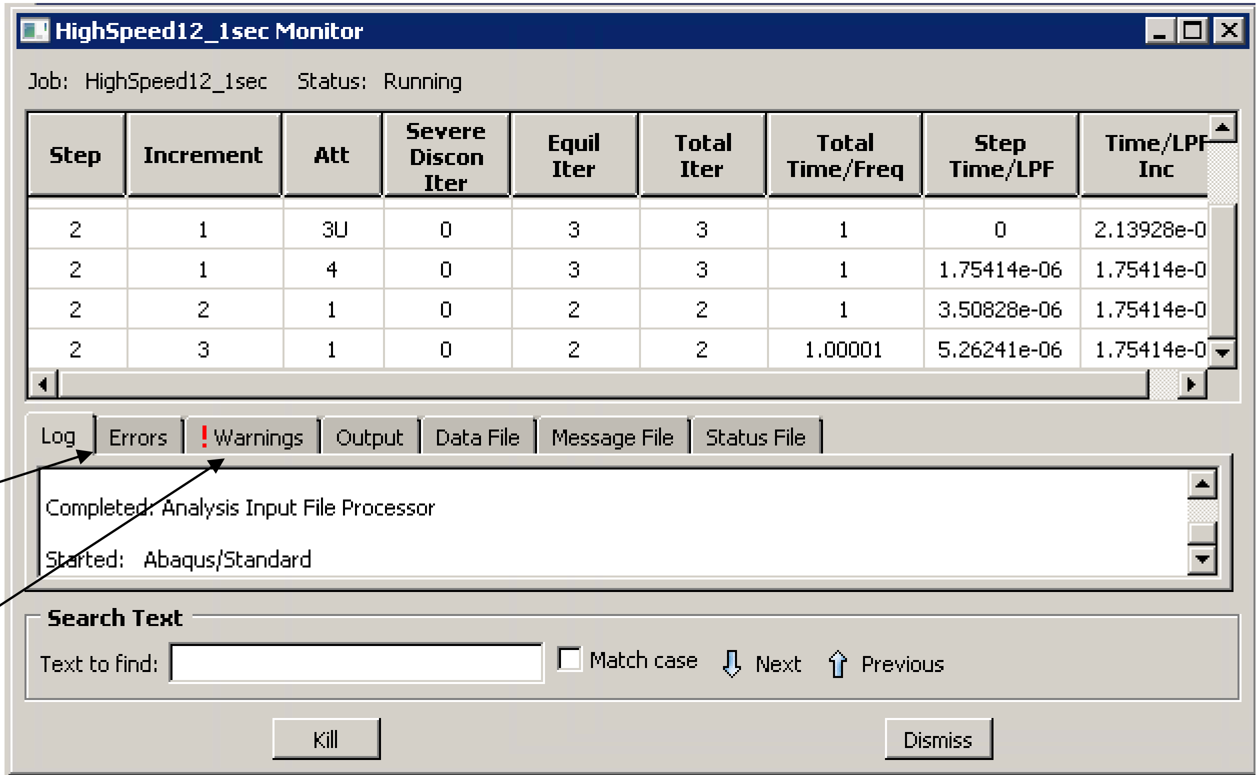


Options:

- **"Write Input"** creates an input file without running the job. This can be used for running a job in the Batch System.
- **"Data Check"** checks for errors in the job, including subroutines. It is useful for checking that a subroutine compiles can be compiled.
- **"Submit"** actually runs the job (creating an input file and a data check are both included in the submitting process).
- **"Monitor"** can be used to check the progress of a running job or see messages produced during an already completed job.
- **"Results"** opens the results file of an already completed job.

Monitoring a job

Right-click on the job name and select “Monitor”.



Errors usually result in the simulation being terminated.

Warnings are things the user should be aware of that might cause problems.

The data file (file extension .dat), message file (.msg), and status file (.sta) can be monitored here. These files are saved to the working directory and can be viewed in a text editor.

Structure

Agenda

Backup

Procedures of finite element analyses in thermal simulation