



AGH

AKADEMIA GÓRNICZO-HUTNICZA
IM. STANISŁAWA STASZICA W KRAKOWIE

ABAQUS

Thermal analysis

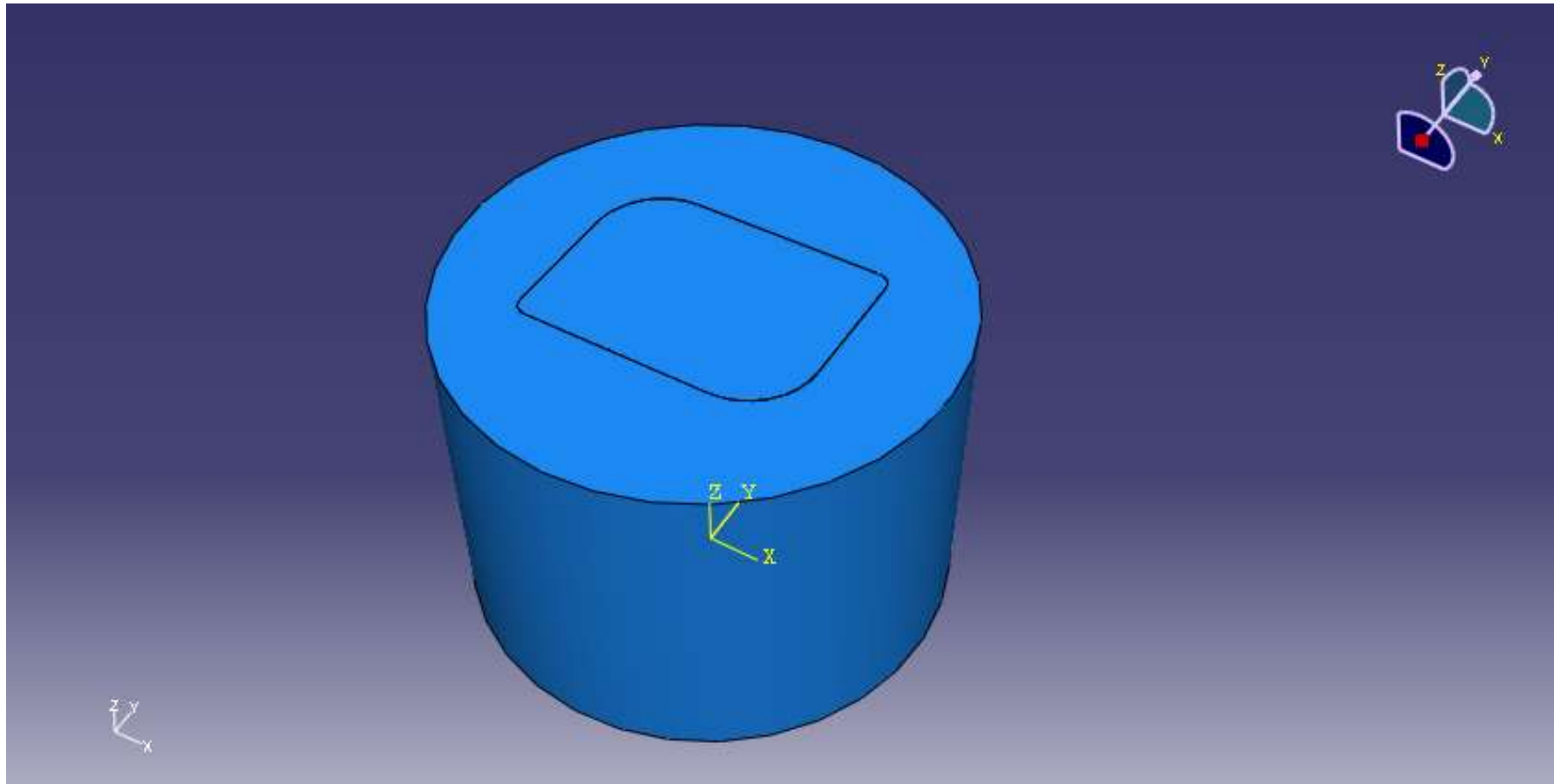
Two steps analysis (thermal and mechanical)

dr inż. Piotr Kustra

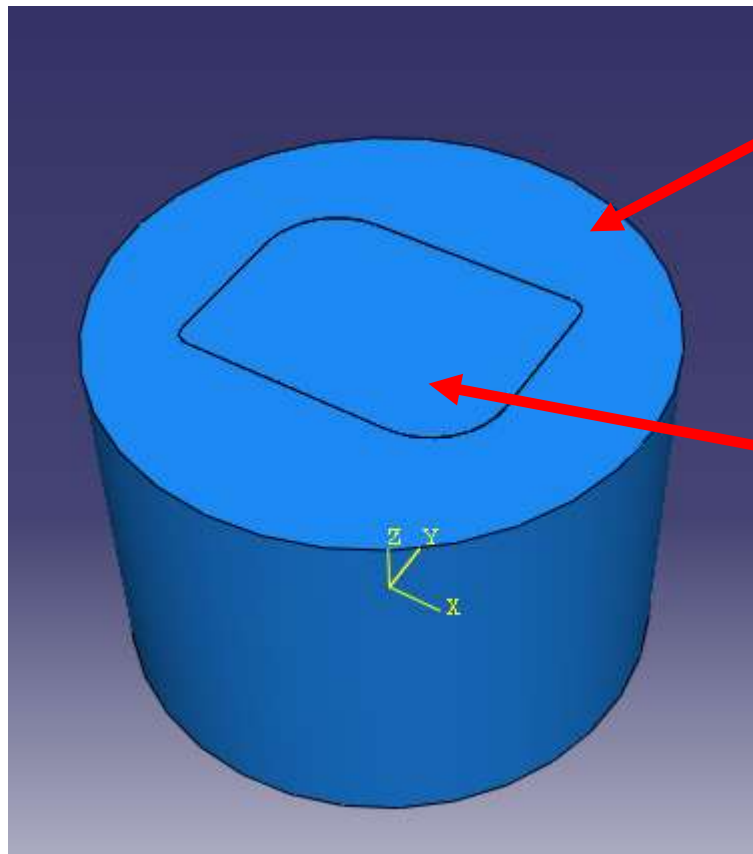
Wydział Inżynierii Metali i Informatyki Przemysłowej



Model definition



Materials data



Resin

$E=13\text{GPa}$

$n=0.35$

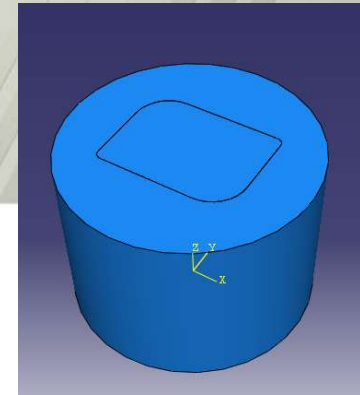
Aluminium

$E=70\text{GPa}$

$n=0.38$



Thermal stress



Materials data m

Resin

Specific heat	1000 J/(Kg K)
Thermal conductivity	1 W·/(m·K)
Expansion	2.3 10 ⁻⁵ 1/C
Density	1719 kg·m ³

Aluminium

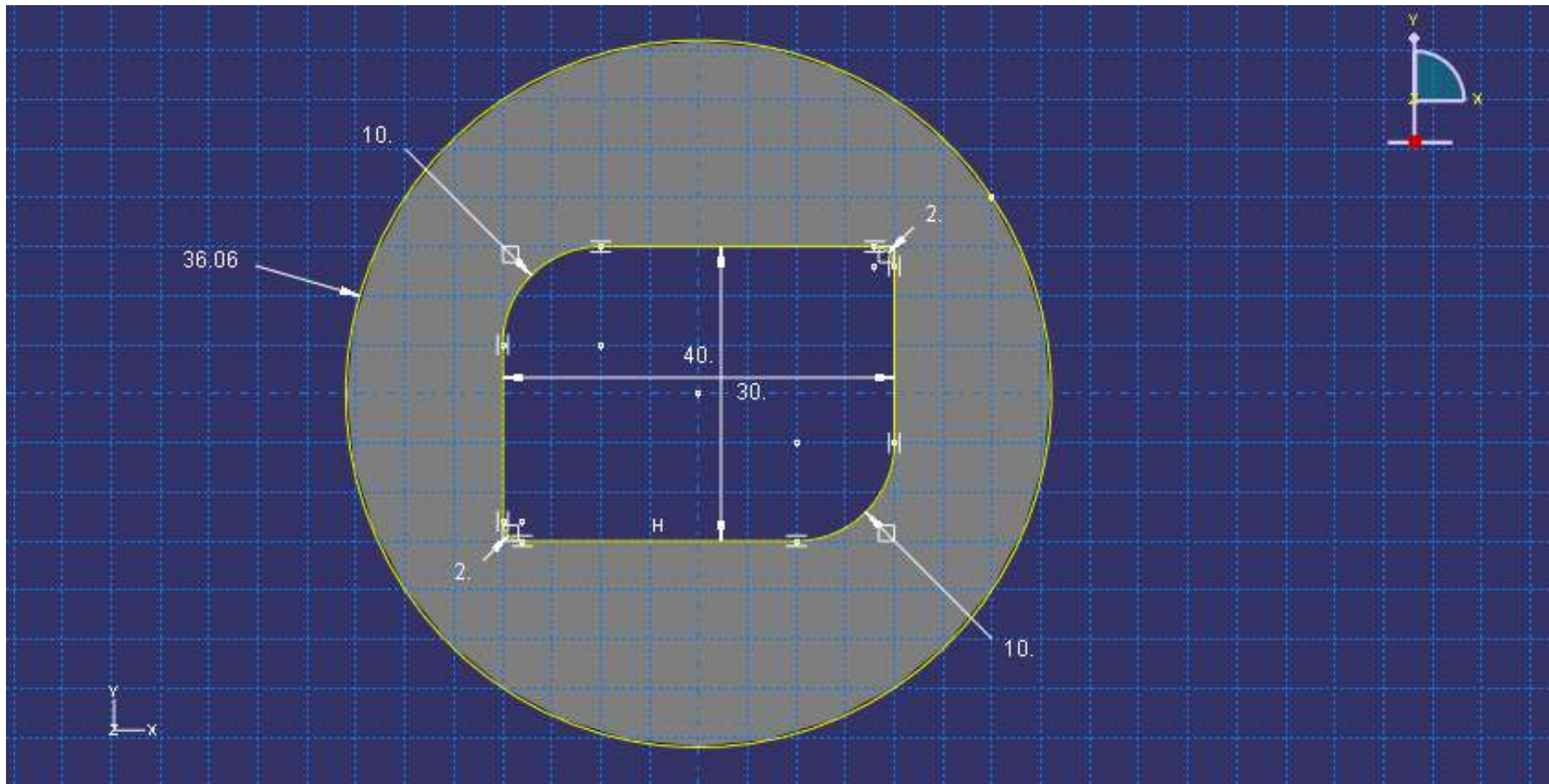
Specific heat	871 J/(Kg K)
Thermal conductivity	202 W·/(m·K)
Expansion	2.3 10 ⁻⁵ 1/C
Density	2700 kg·m ³

Data for ABAQUS (model in mm)

Specific heat	1000000000
Thermal conductivity	1
Expansion	4.2E-005
Density	1.719E-009

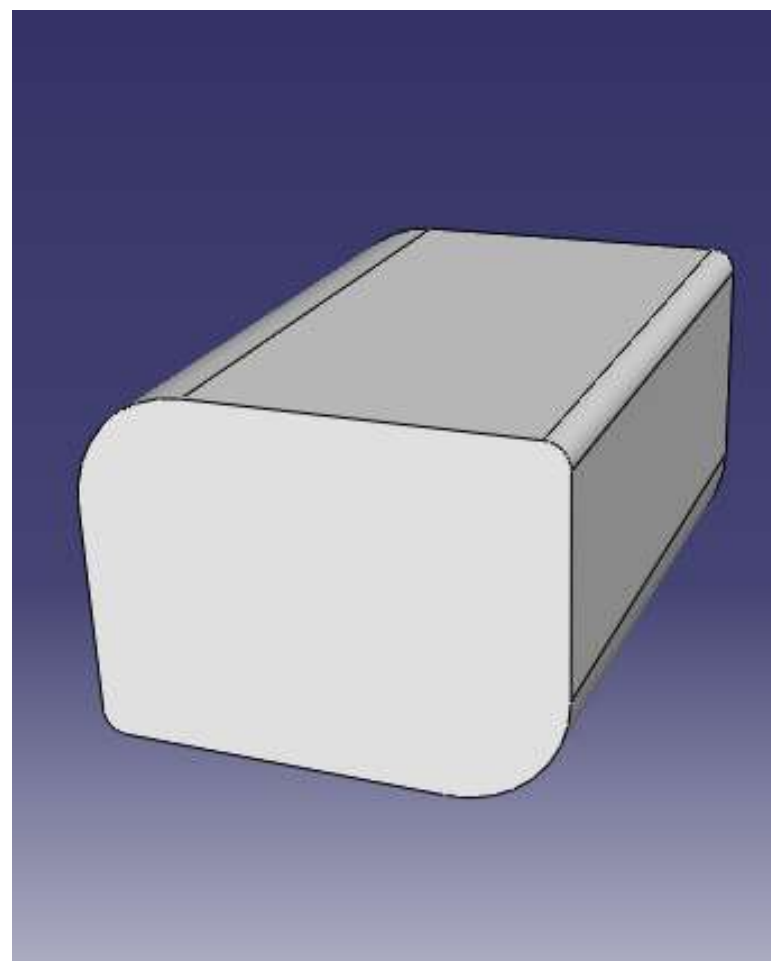
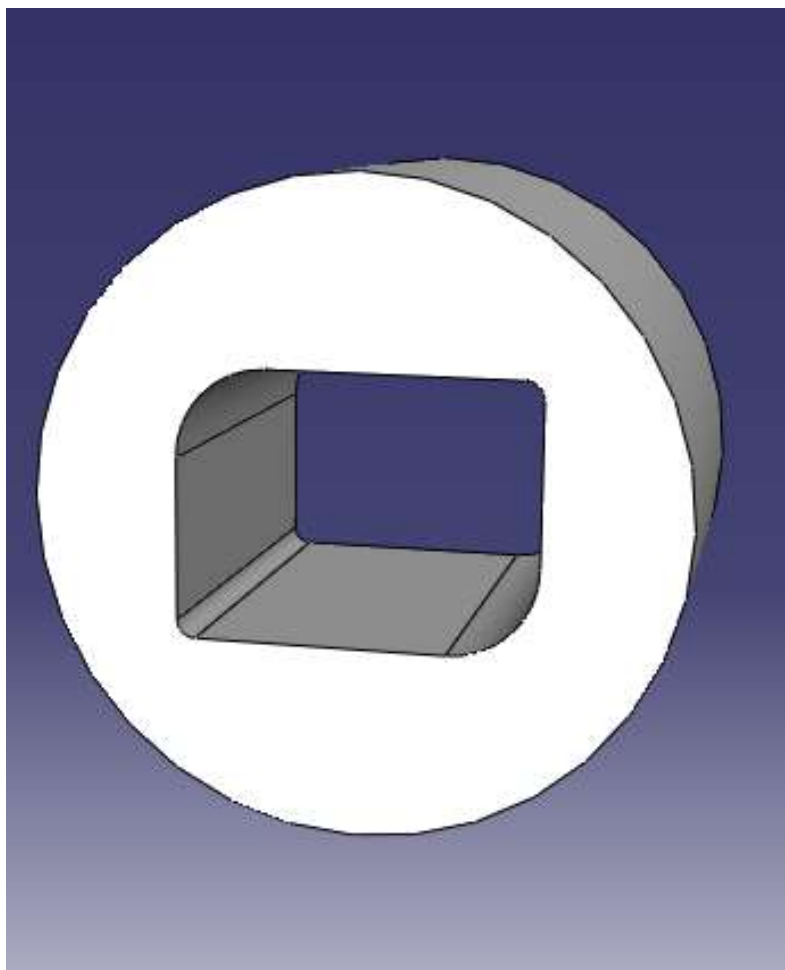
Specific heat	871000000
Thermal conductivity	202
Expansion	2.3E-005
Density	2.7E-009

Geometry definition





Parts





Analysis type definition

The screenshot displays the Abaqus/CAE interface with the 'Edit Step' dialog box open. The dialog is configured for a 'Heat transfer' analysis of type 'Incrementation'. The 'Type' is set to 'Automatic'. The 'Maximum number of increments' is 10000. The 'Increment size' is 0.1, with a 'Minimum' of 2E-005 and a 'Maximum' of 30. The 'End step when temperature change is less than' checkbox is checked, but the value is obscured by a blue box. The 'Max. allowable temperature change per increment' is 2, and the 'Max. allowable emissivity change per increment' is 0.1.

	Initial	Minimum	Maximum
Increment size:	0.1	2E-005	30



Contact definition

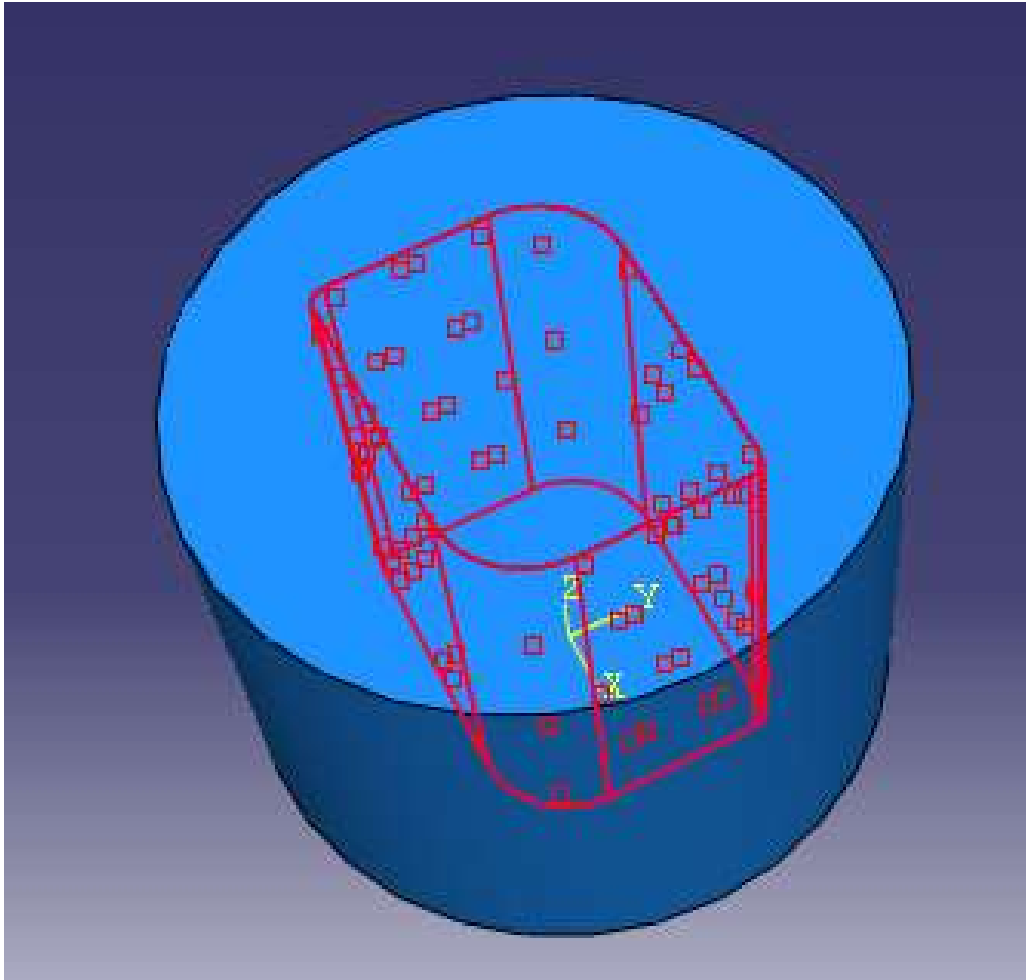
The screenshot displays the Abaqus/CAE interface with the 'Edit Interaction' dialog box open. The dialog box is titled 'Edit Interaction' and contains the following settings:

- Name: Int-1
- Type: Surface-to-surface contact (Standard)
- Step: Initial
- Master surface: (Picked) Edit Region... [Red square]
- Slave surface: (Picked) Edit Region... [Magenta square] [Switch]
- Sliding formulation: Finite sliding Small sliding
- Discretization method: Surface to surface [Dropdown]
- Exclude shell/membrane element thickness
- Degree of smoothing for master surface: 0.2 [Text box]
- Use supplementary contact points: Selectively Never Always
- Constraint position: Node centered Face centered
- Contact tracking: Single configuration (state) Two configurations (path)
- Slave Node/Surface Adjustment: [Clearance] [Text box]
- No adjustment
- Adjust only to remove overclosure
- Specify tolerance for adjustment zone: 0 [Text box]
- Adjust slave nodes in set: [Dropdown]
- Contact interaction property: IntProp-1 [Dropdown] [Create...]

The background shows the Abaqus/CAE software interface with the 'Model Database' tree on the left, the 'Interaction' module selected, and a status bar at the bottom with error messages.



Thermal Conductance



Edit Contact Property

Name: IntProp-1

Contact Property Options

Thermal Conductance

Mechanical Thermal Delete

Thermal Conductance

Definition: Tabular

Use only clearance-dependency data
 Use only pressure-dependency data
 Use both clearance- and pressure-dependency data

Clearance Dependency Pressure Dependency

Use temperature-dependent data
 Use mass flow rate-dependent data (Standard only)

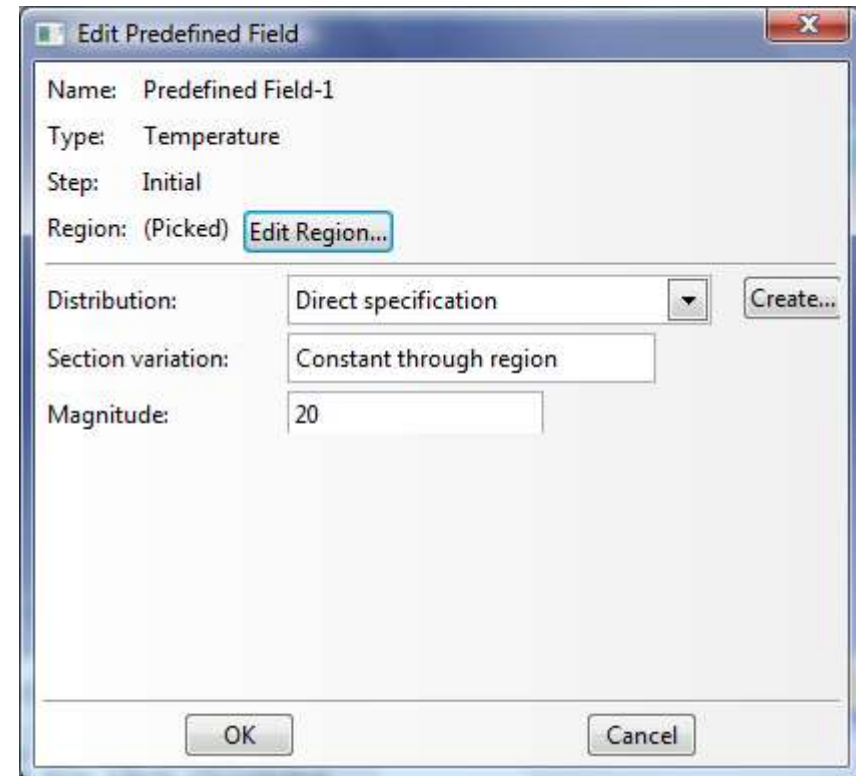
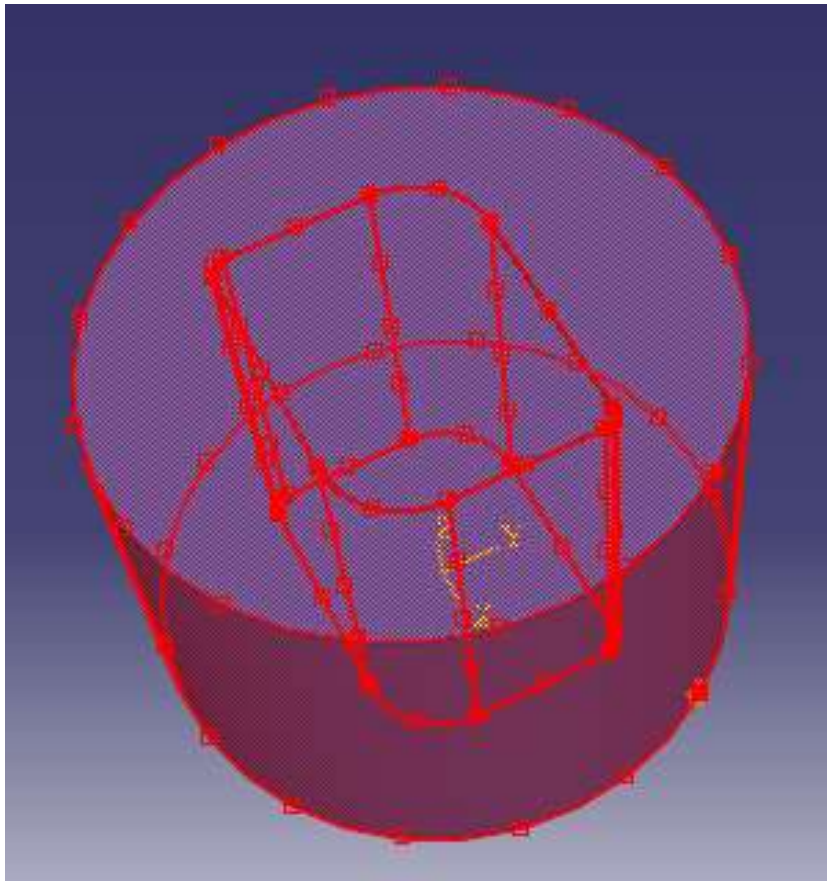
Number of field variables: 0

Conductivity	Clearance
0.202	0
0	1

OK Cancel



Initial temperature





Surface film condition

The screenshot displays the Abaqus software interface. The main viewport shows a 3D model of a red sphere with a surface film condition being defined. The 'Edit Interaction' dialog box is open, showing the following settings:

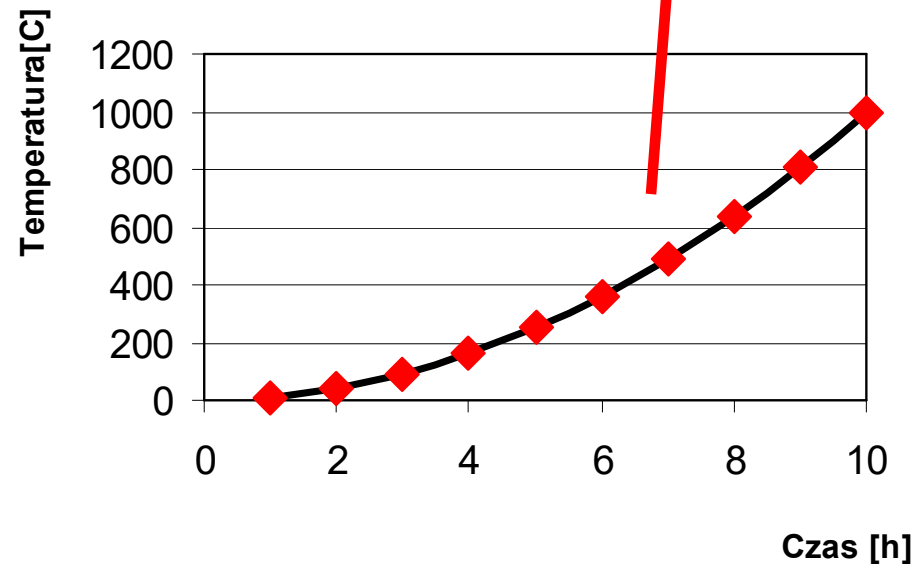
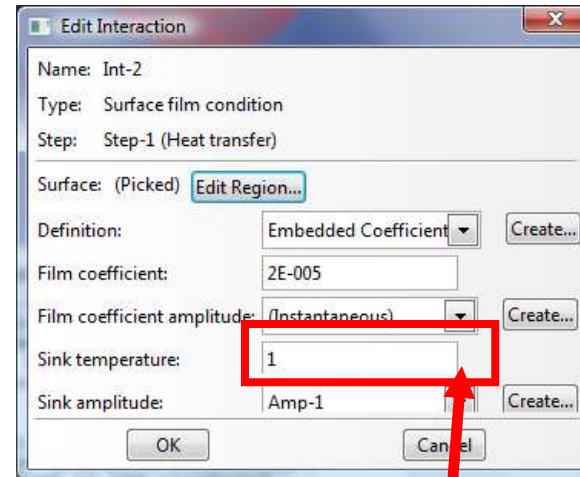
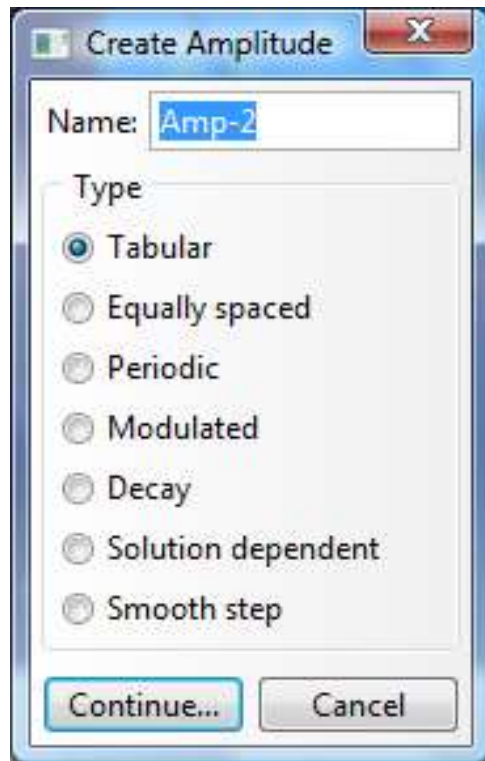
- Name: Int-2
- Type: Surface film condition
- Step: Step-1 (Heat transfer)
- Surface: (Picked) [Edit Region...](#)
- Definition: Embedded Coefficient [Create...](#)
- Film coefficient: 2E-005
- Film coefficient amplitude: (Instantaneous) [Create...](#)
- Sink temperature: 1
- Sink amplitude: Amp-1 [Create...](#)

The bottom status bar shows the following text:

```
The contents of viewport "Viewport: 1" have been copied to the clipboard.
Error in job mechanikapotermice: Too many attempts made for this job.
Job mechanikapotermice: Abaqus/Standard aborted due to errors.
Error in job mechanikapotermice: Abaqus/Standard Analysis exited with
code = 1.
file for possible error messages if the file exists.
Job mechanikapotermice aborted due to errors.
The job input file "mechanikapotermice.inp" has been submitted.
Job mechanikapotermice: Analysis Input File Processor completed.
```

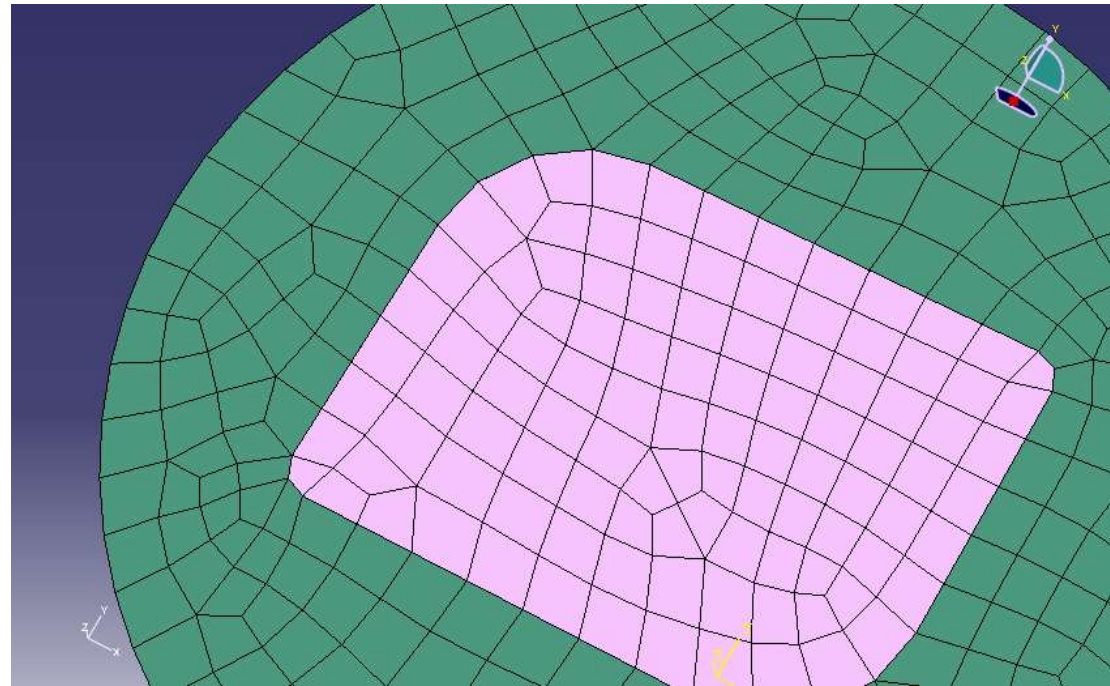
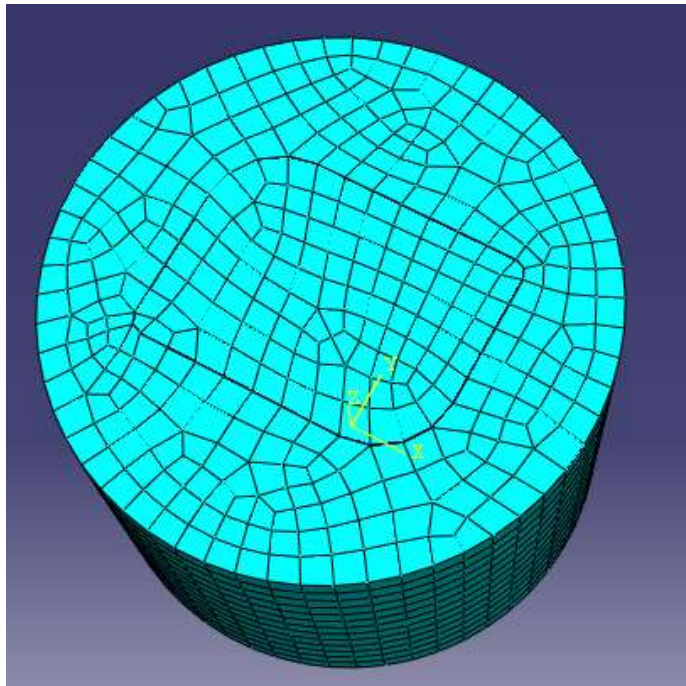
Outside temperature - heating

Amplitude





FEM grids





FEM element type

