



AGH

AKADEMIA GÓRNICZO-HUTNICZA
IM. STANISŁAWA STASZICA W KRAKOWIE

ABAQUS

Thermal analysis

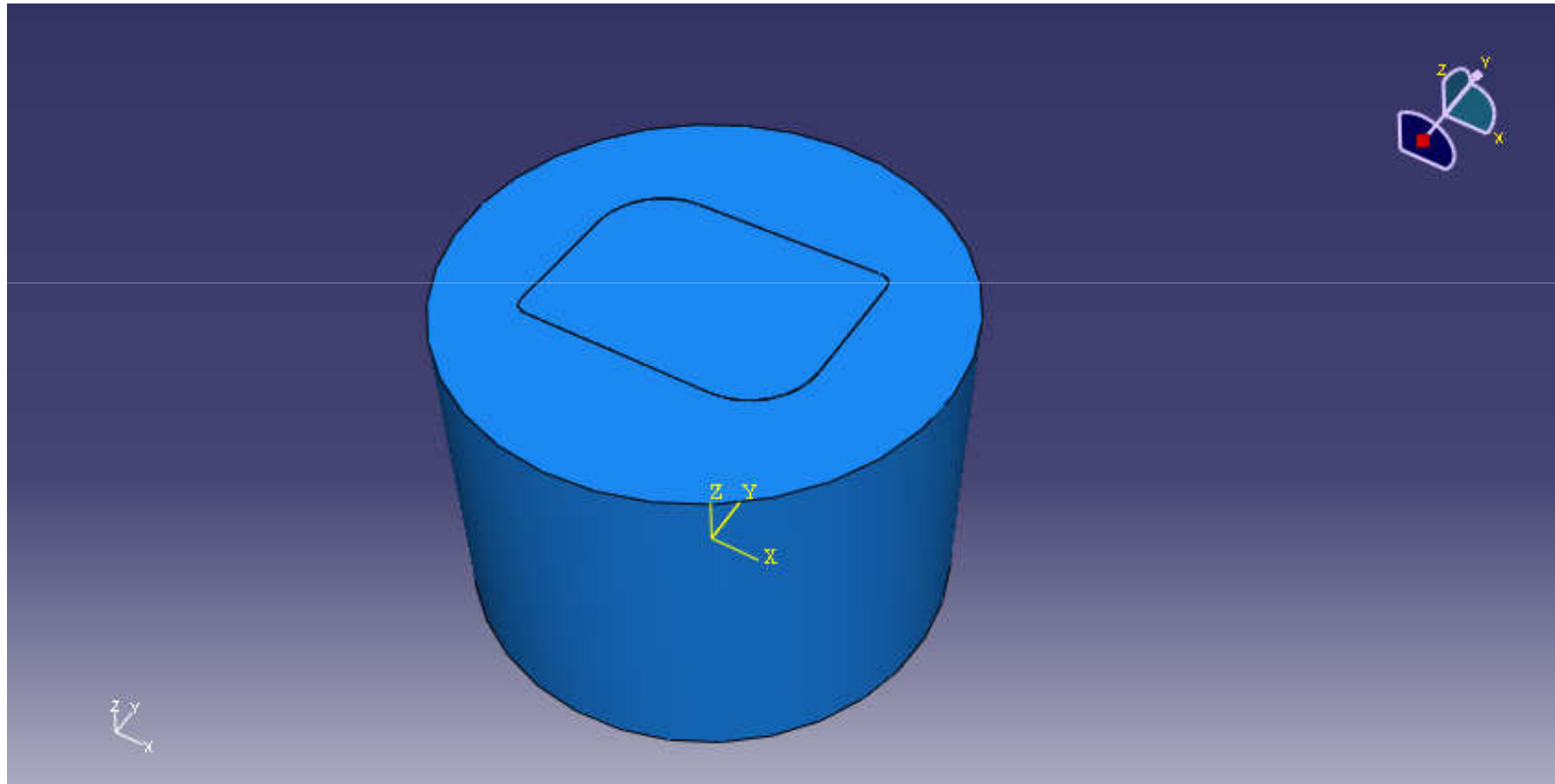
Two steps analysis (thermal and mechanical)

dr inż. Piotr Kustra

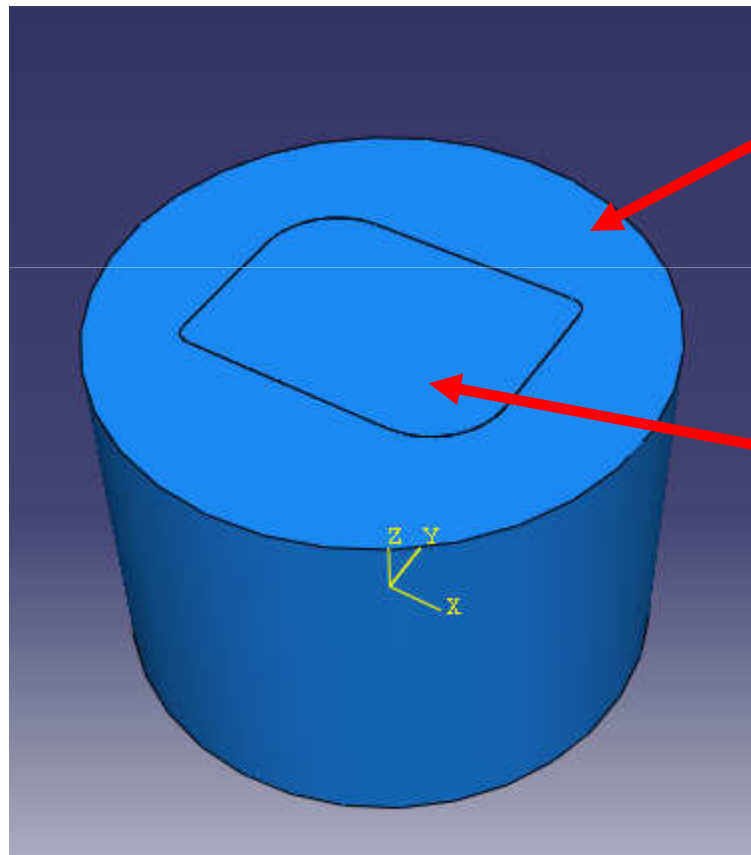
Wydział Inżynierii Metali i Informatyki Przemysłowej



Model definition



Materials data



Resin

$E=13\text{GPa}$

$n=0.35$

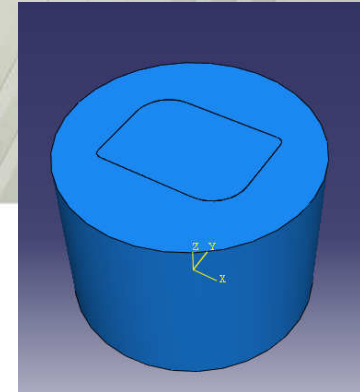
Aluminium

$E=70\text{GPa}$

$n=0.38$



Thermal stress



Materials data

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

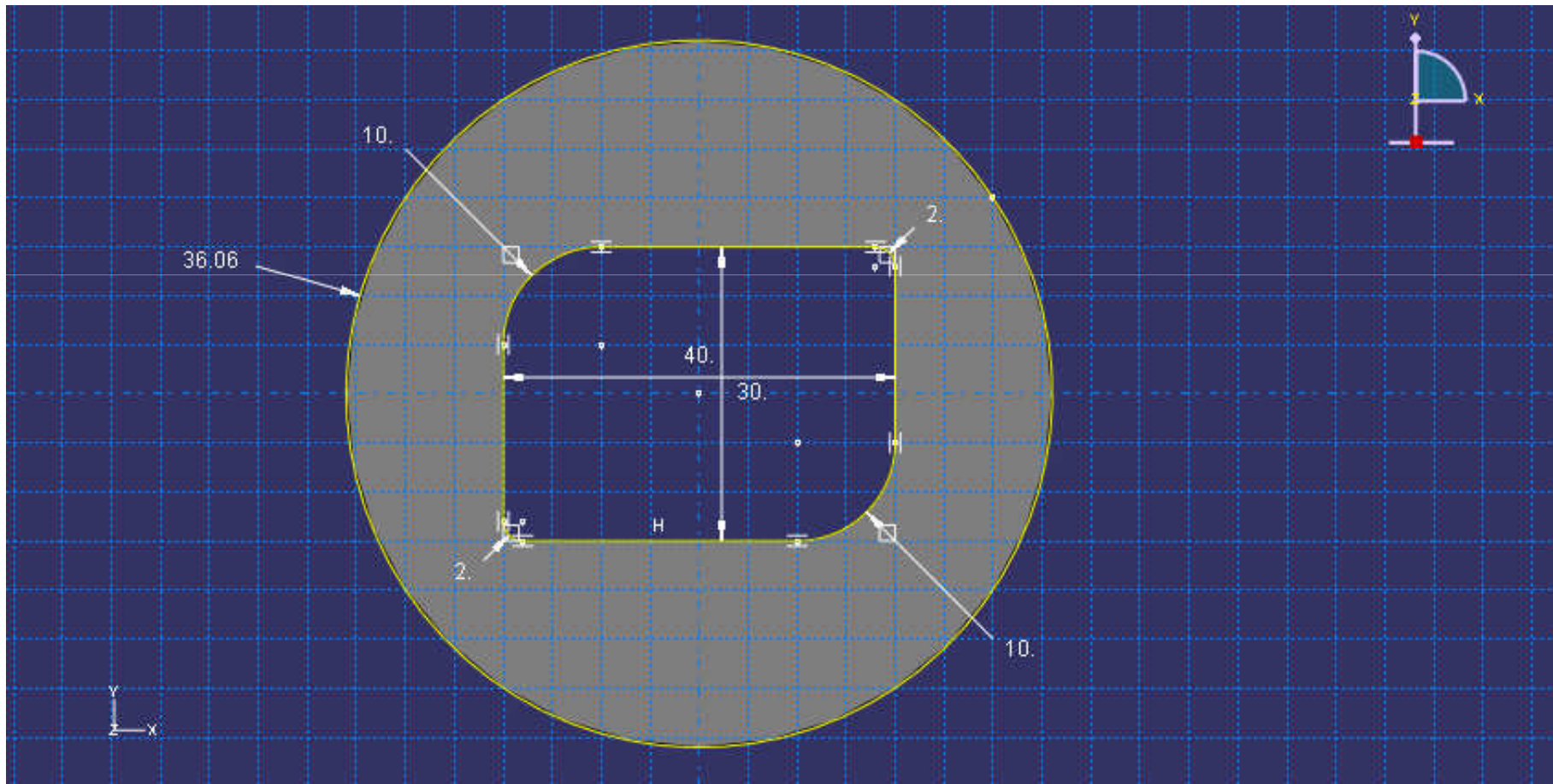
Aluminium

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)

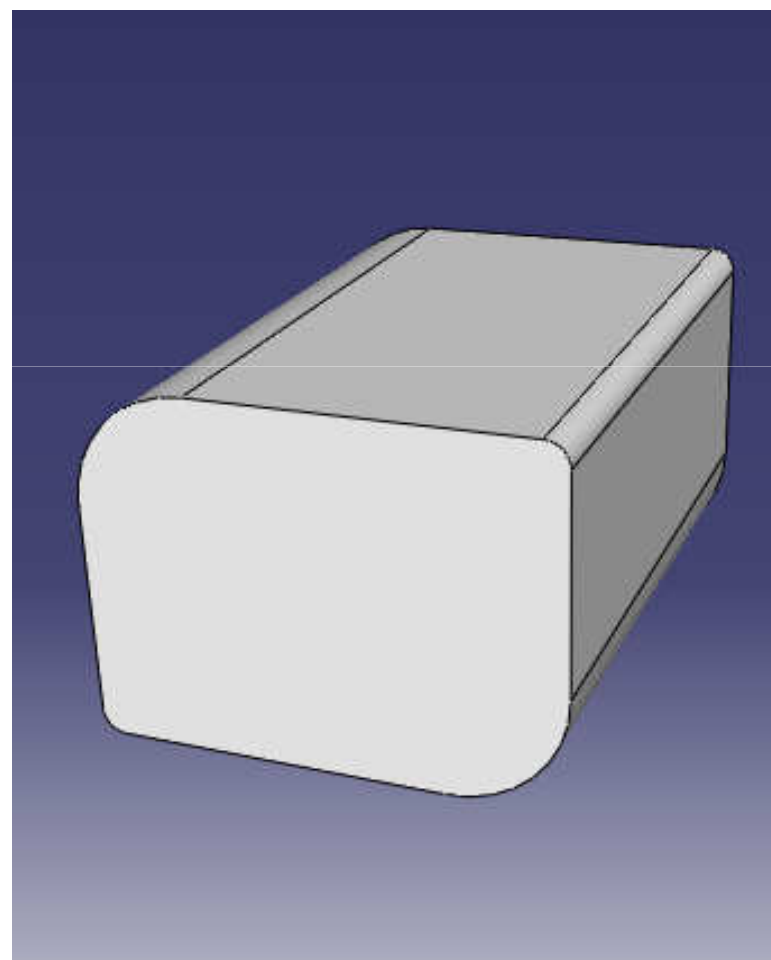
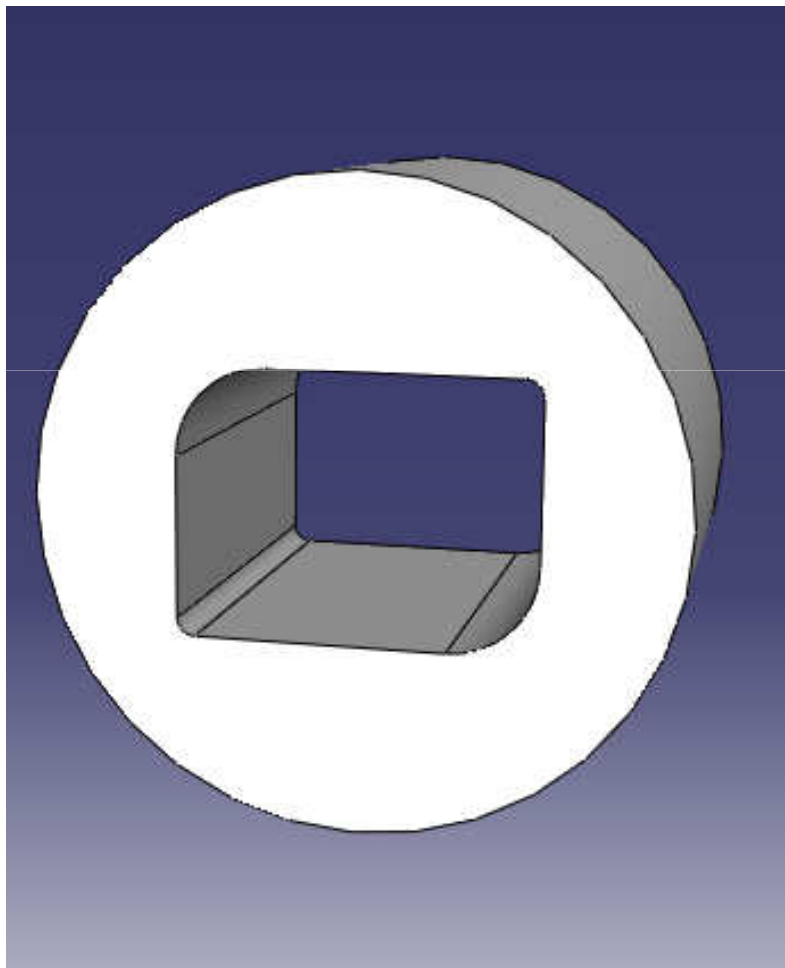
Specific heat	1000000000
Thermal conductivity	1
Expansion	4.2E-005
Density	1.719E-009

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





Parts





Analysis type

The screenshot displays the Abaqus CAE interface with the 'Edit Step' dialog box open. The dialog box is titled 'Edit Step' and shows the following configuration for 'Step-1':

- Name: Step-1
- Type: Heat transfer
- Tab: Incrementation
- Type: Automatic Fixed
- Maximum number of increments: 10000
- Increment size table:

	Initial	Minimum	Maximum
Increment size:	0.1	2E-005	30
- End step when temperature change is less than: []
- Max. allowable temperature change per increment: 2
- Max. allowable emissivity change per increment: 0.1

The background shows the Abaqus Model Database tree with 'Step-1' selected under 'Model-3'.



Contact definition

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

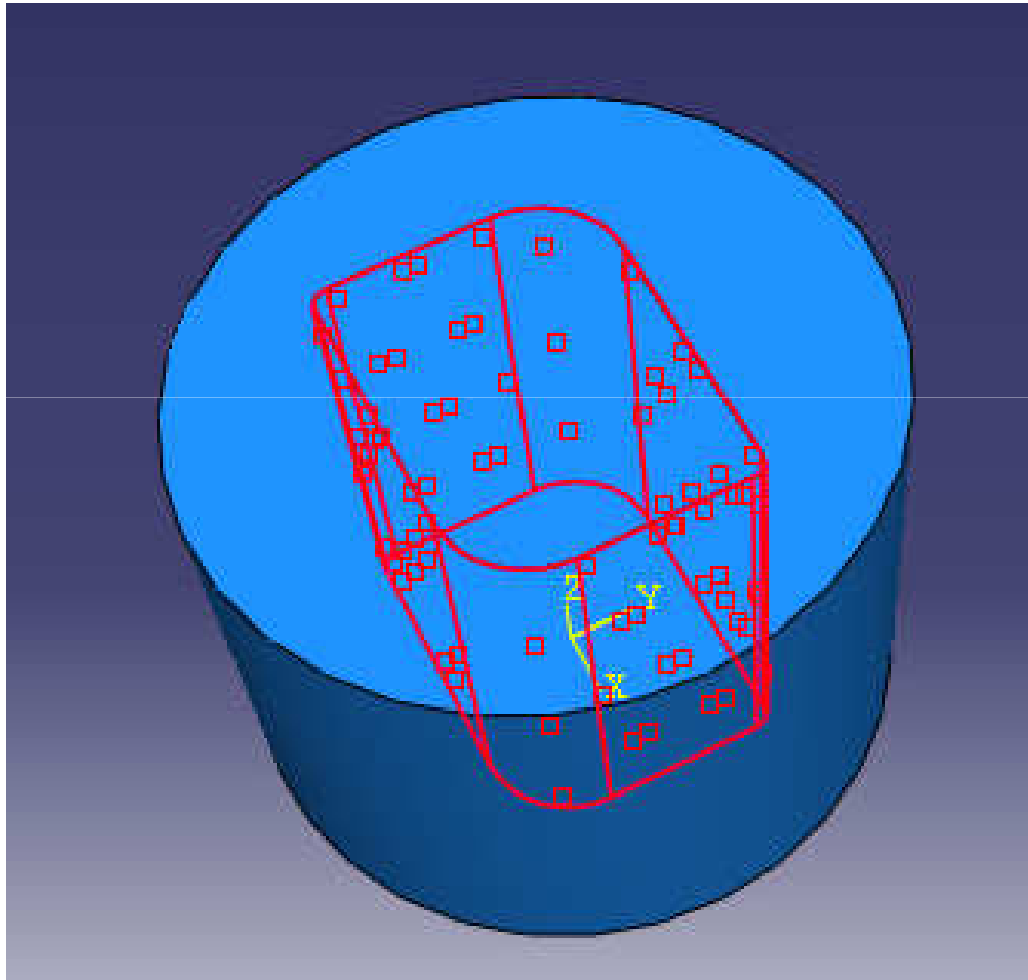
- Name: Int-1
- Type: Surface-to-surface contact (Standard)
- Step: Initial
- Master surface: (Picked) Edit Region... [Red square]
- Slave surface: (Picked) Edit Region... [Magenta square] [Switch]
- Sliding formulation: Finite sliding Small sliding
- Discretization method: Surface to surface [Dropdown]
- 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 main window with the 'Model Database' tree on the left, the 'Interaction' module selected, and a toolbar at the top. The status bar at the bottom contains the following text:

The contents of viewport "Viewport: 1" have been copied.
The contents of viewport "Viewport: 1" have been copied.
The contents of viewport "Viewport: 1" have been copied.
Error in job mechanikapoter.mic: Too many attempts made to create the job.



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)

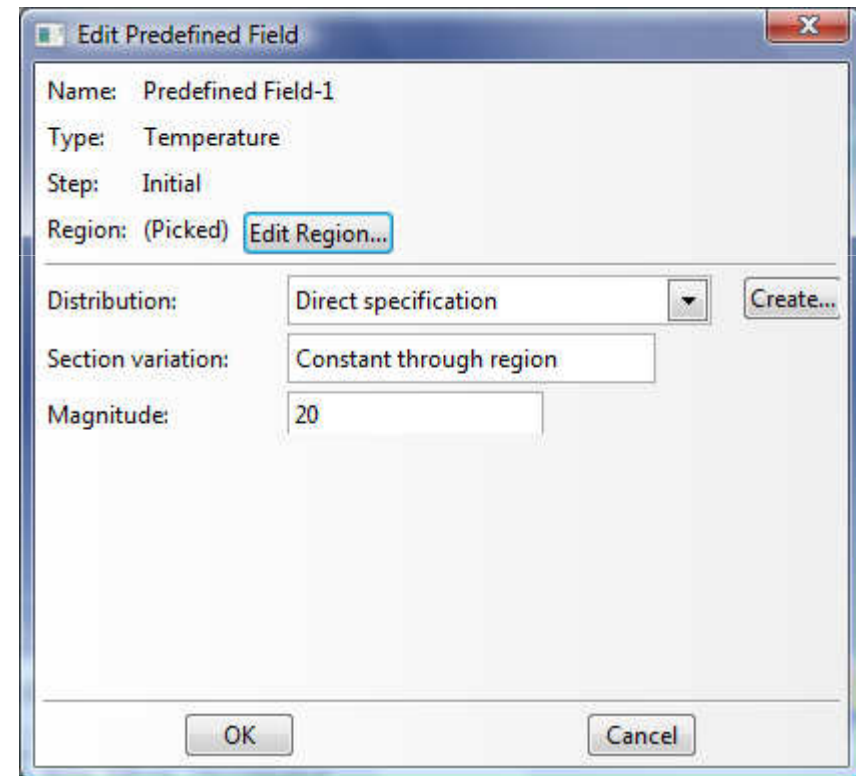
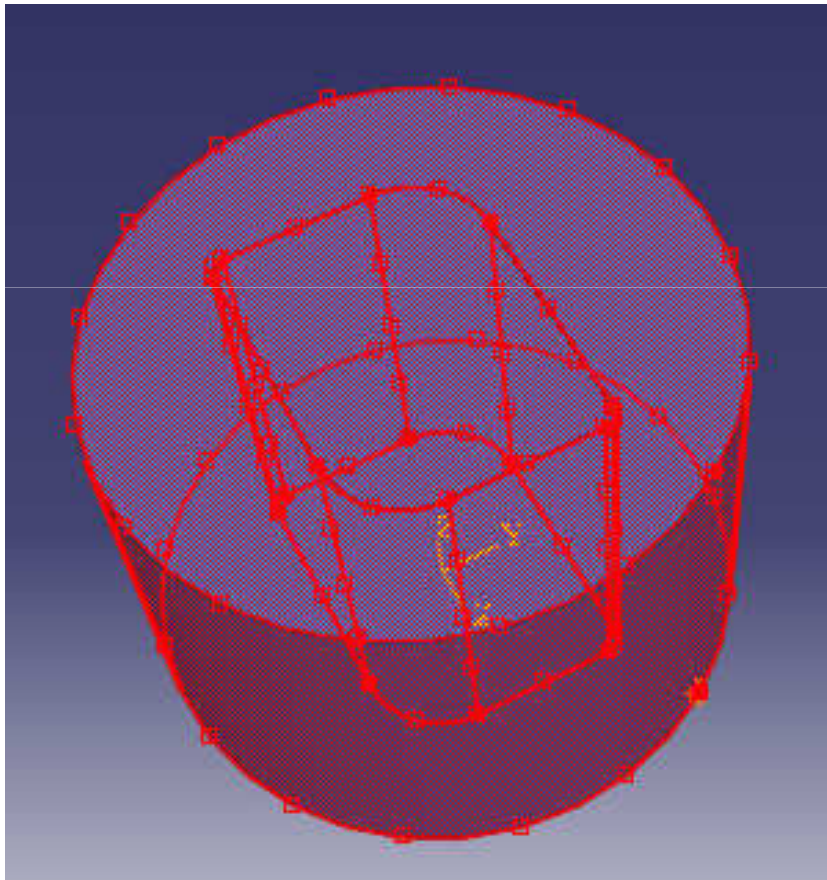
Number of field variables: 0

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 mechanical part with a red surface film condition applied to a specific region. The 'Edit Interaction' dialog box is open, showing the following settings:

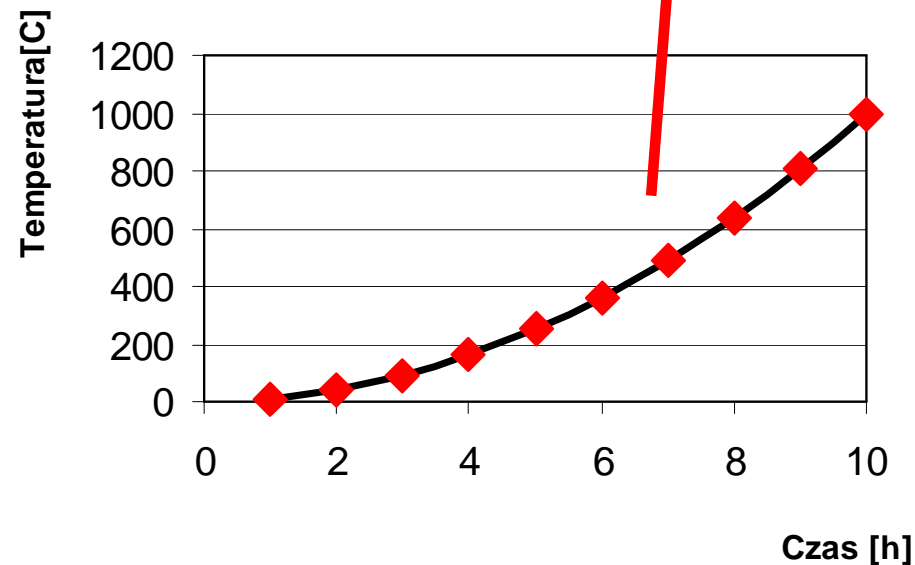
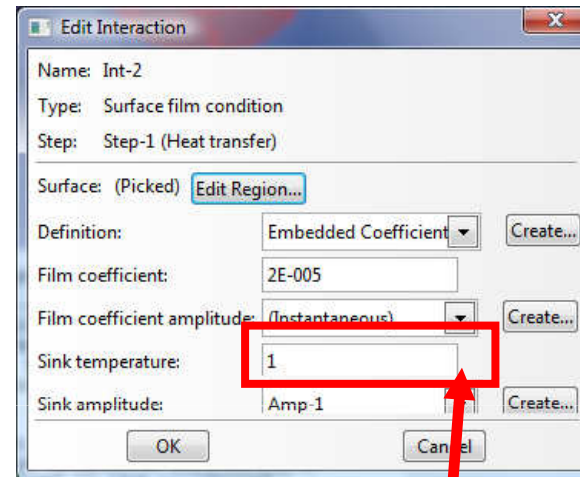
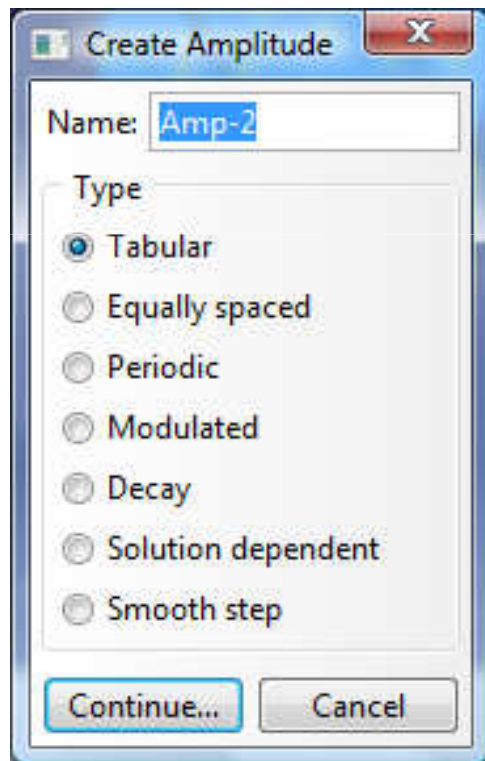
- Name: Int-2
- Type: Surface film condition
- Step: Step-1 (Heat transfer)
- Surface: (Picked) [Edit Region...](#)
- Definition: Embedded Coefficient [Create...](#)
- Film coefficient: 2E-005
- Film coefficient amplitude: (Instantaneous) [Create...](#)
- Sink temperature: 1
- Sink amplitude: Amp-1 [Create...](#)

The bottom status bar contains 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
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.
```

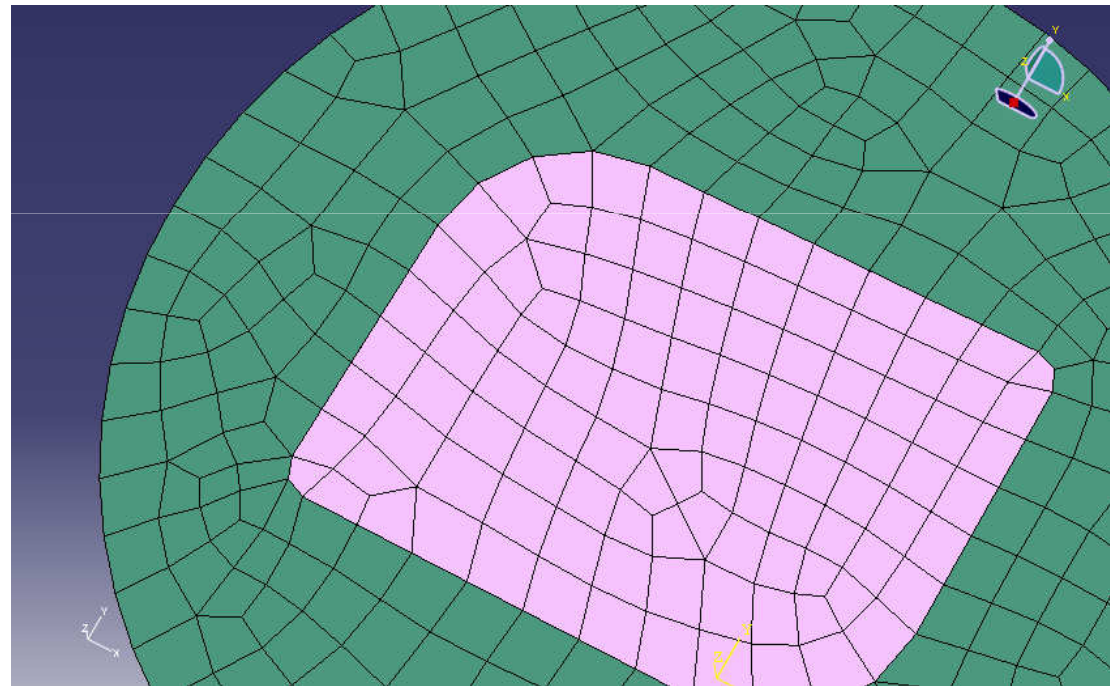
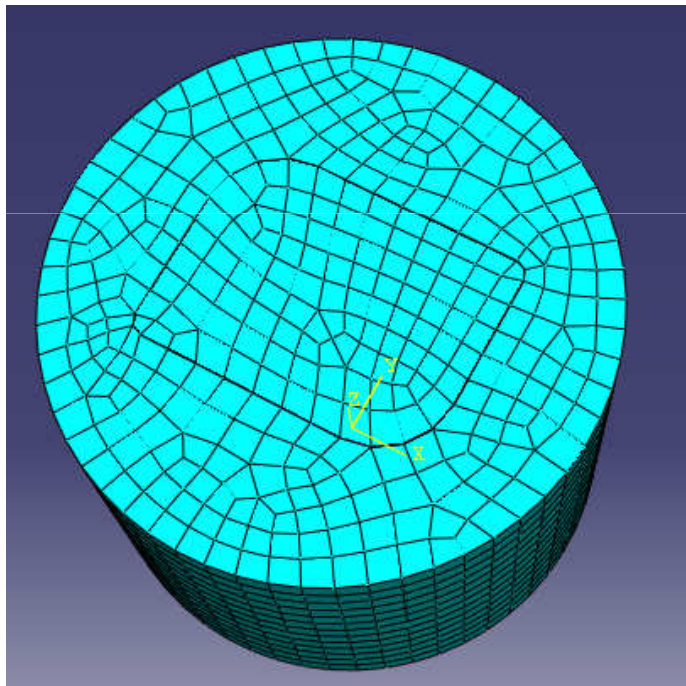
Outside temperature - heating

Amplitude



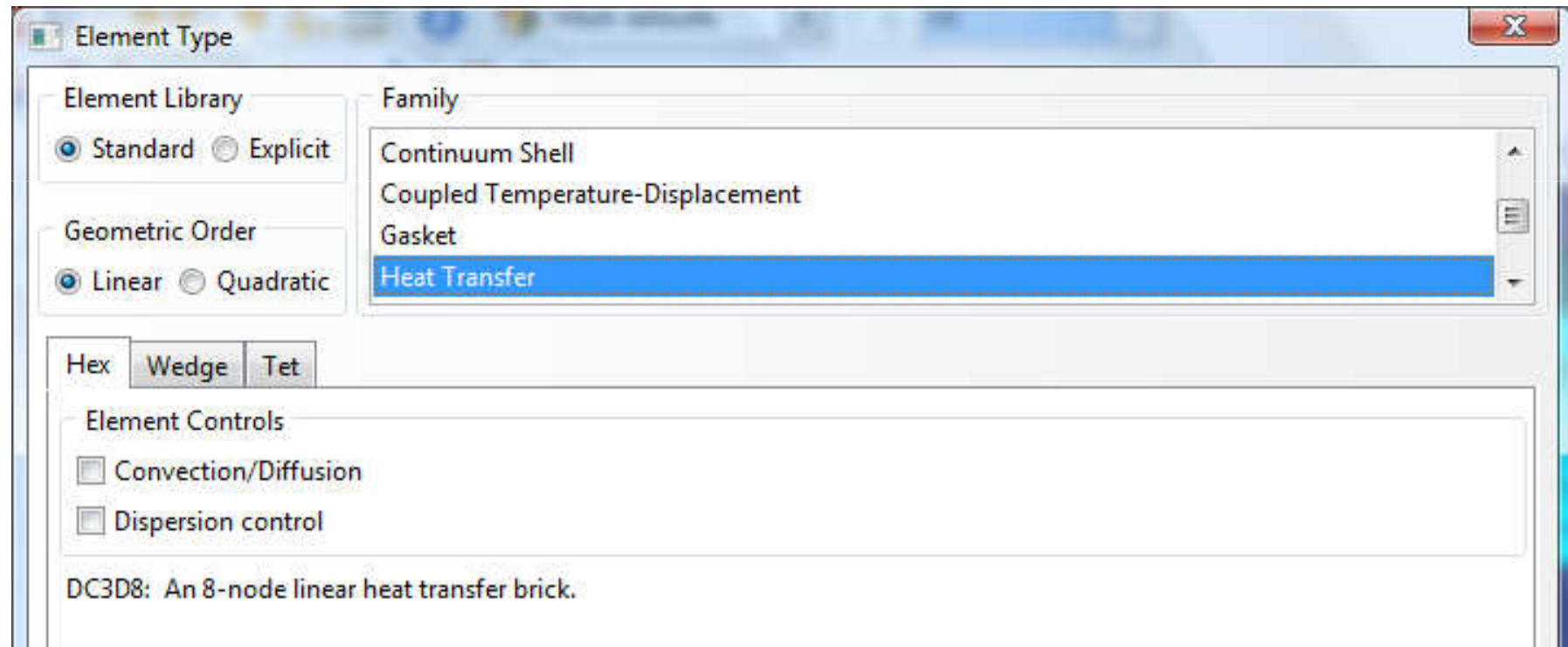


FEM grids



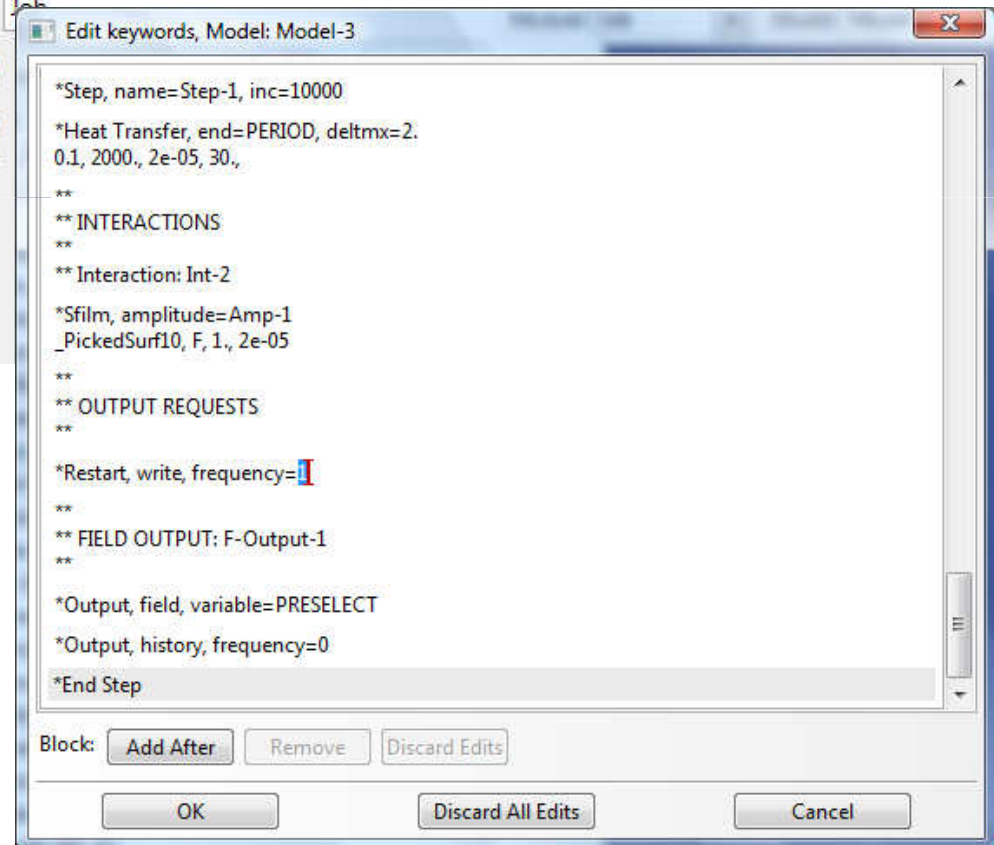
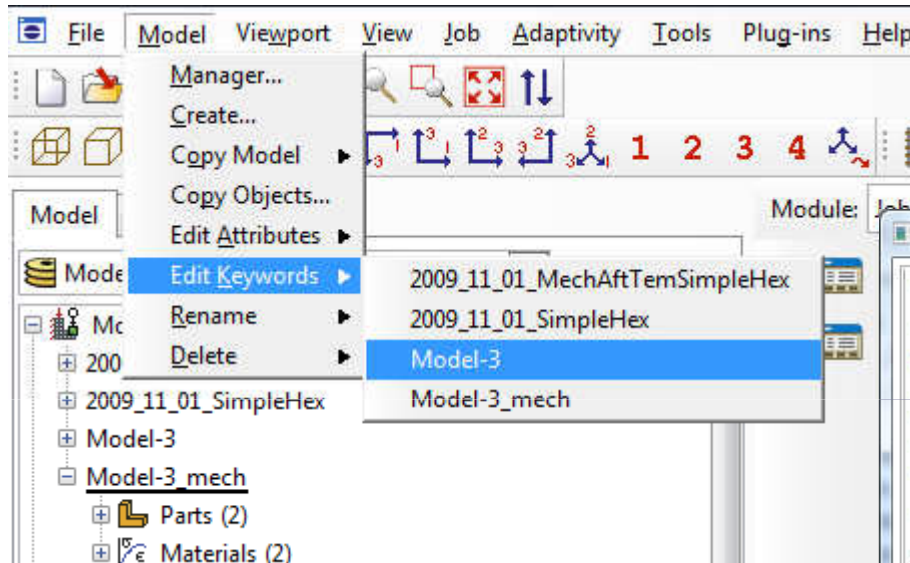


FEM element type





Two steps analysis – thermal and mechanical



For two-staged analysis - thermal and mechanical

In thermal analysis user needs to edit Keywords :

Restart, write, frequency = 1



Mechanical analysis

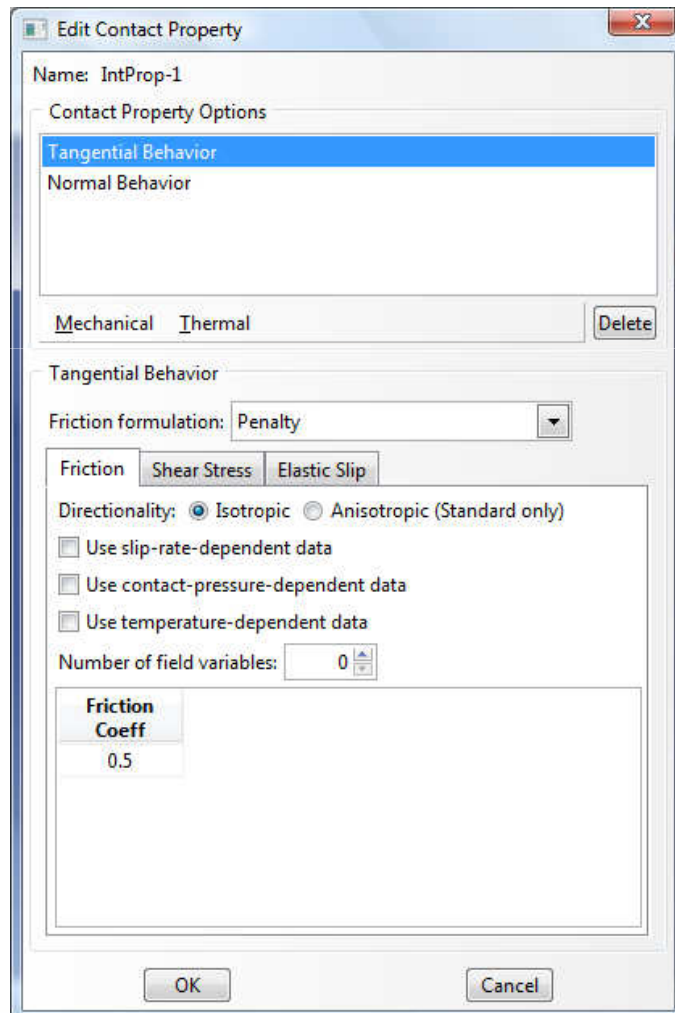
The screenshot displays the ANSYS Workbench interface for a mechanical analysis. The main window shows a 3D model of a part with a contact interaction defined. The 'Edit Interaction' dialog box is open, showing the following settings:

- Name: Int-1
- Type: Surface-to-surface contact (Standard)
- Step: Initial
- Master surface: (Picked) Edit Region... [Red square]
- Slave surface: (Picked) Edit Region... [Purple square] [Switch]
- Sliding formulation: Finite sliding Small sliding
- Discretization method: Surface to surface
- Exclude shell/membrane element thickness
- Degree of smoothing for master surface: 0,2
- 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
- No adjustment
- Adjust only to remove overclosure
- Specify tolerance for adjustment zone: 0
- Adjust slave nodes in set: [Dropdown]
- Contact interaction property: IntProp-1 [Create...]
- Options: Interference Fit...
- Contact controls: (Default) [Dropdown]

The 'Model Database' on the left shows the following structure:

- Model-3_mech
 - Parts (2)
 - Materials (2)
 - Sections (2)
 - Profiles
 - Assembly
 - Steps (2)
 - Initial
 - Interactions (1)
 - Int-1 (Created)
 - BCs (4)
 - BC-2 (Created)
 - BC-3 (Created)
 - BC-4 (Created)
 - BC-5 (Created)
 - Predefined Fields (1)
 - Step-1
 - Field Output Requests (1)
 - F-Output-1 (Created)
 - History Output Requests (1)
 - ALE Adaptive Mesh Constraints
 - Interactions (1)
 - Loads

Analiza mechaniczna



Edit Contact Property

Name: IntProp-1

Contact Property Options

Tangential Behavior

Normal Behavior

Mechanical Thermal Delete

Tangential Behavior

Friction formulation: Penalty

Friction Shear Stress Elastic Slip

Directionality: Isotropic Anisotropic (Standard only)

Use slip-rate-dependent data

Use contact-pressure-dependent data

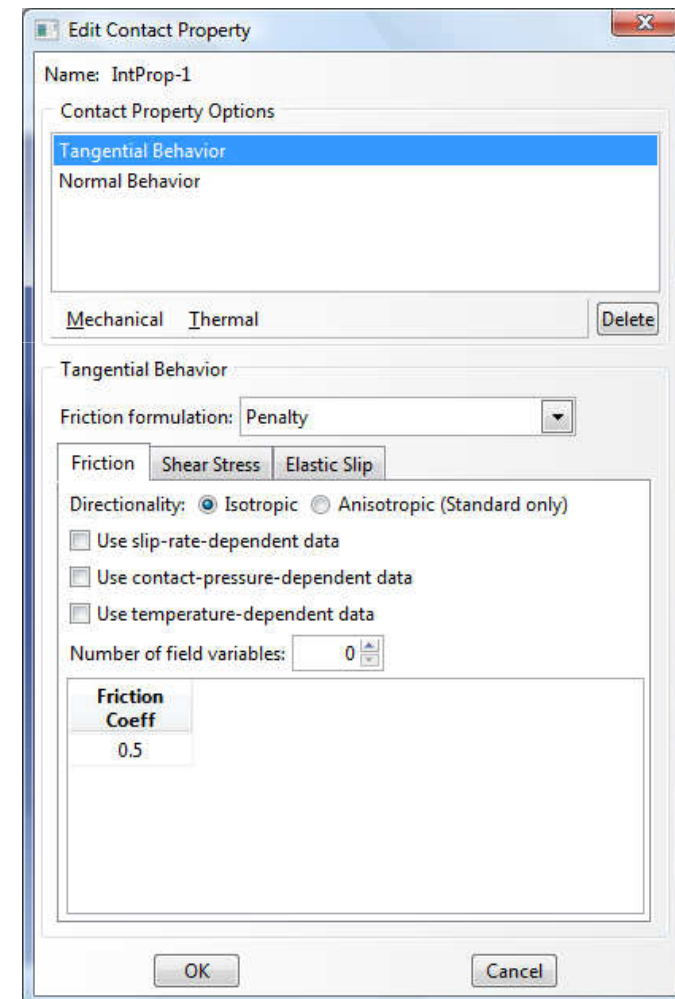
Use temperature-dependent data

Number of field variables: 0

Friction Coeff
0.5

OK Cancel

Analiza termiczna



Edit Contact Property

Name: IntProp-1

Contact Property Options

Tangential Behavior

Normal Behavior

Mechanical Thermal Delete

Tangential Behavior

Friction formulation: Penalty

Friction Shear Stress Elastic Slip

Directionality: Isotropic Anisotropic (Standard only)

Use slip-rate-dependent data

Use contact-pressure-dependent data

Use temperature-dependent data

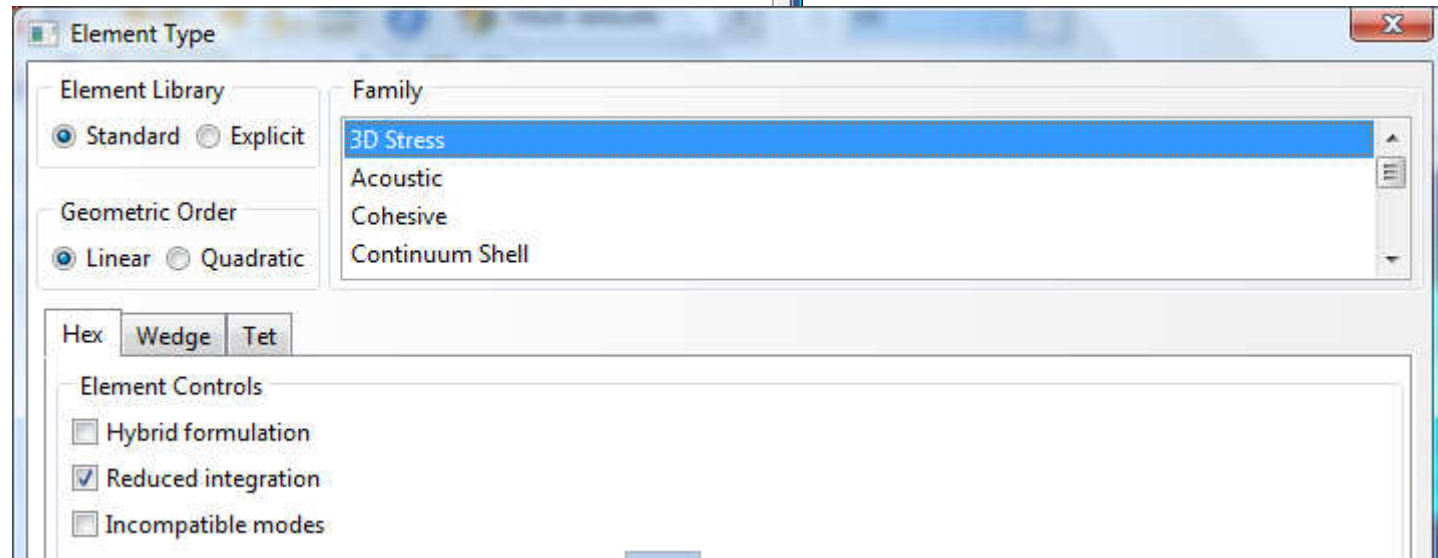
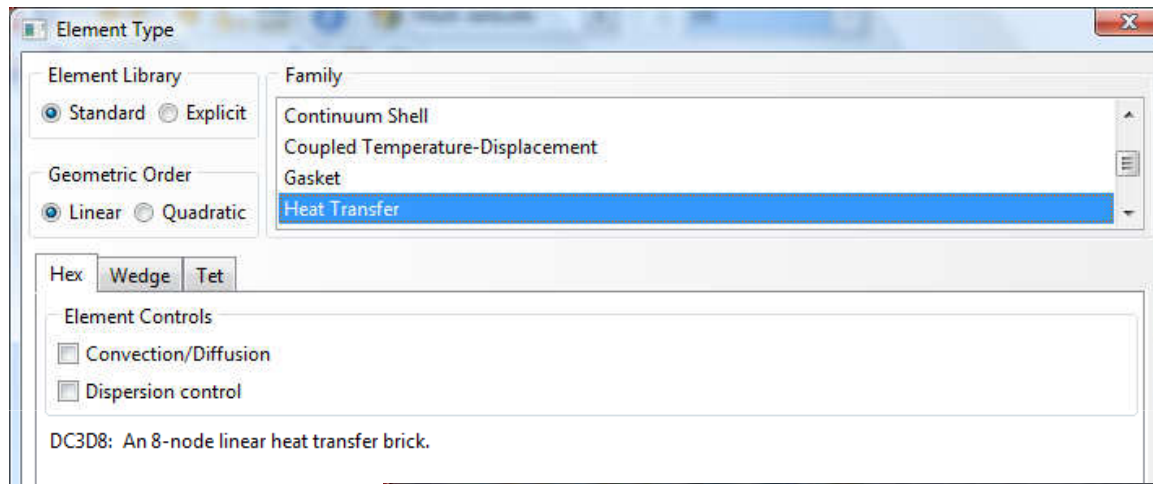
Number of field variables: 0

Friction Coeff
0.5

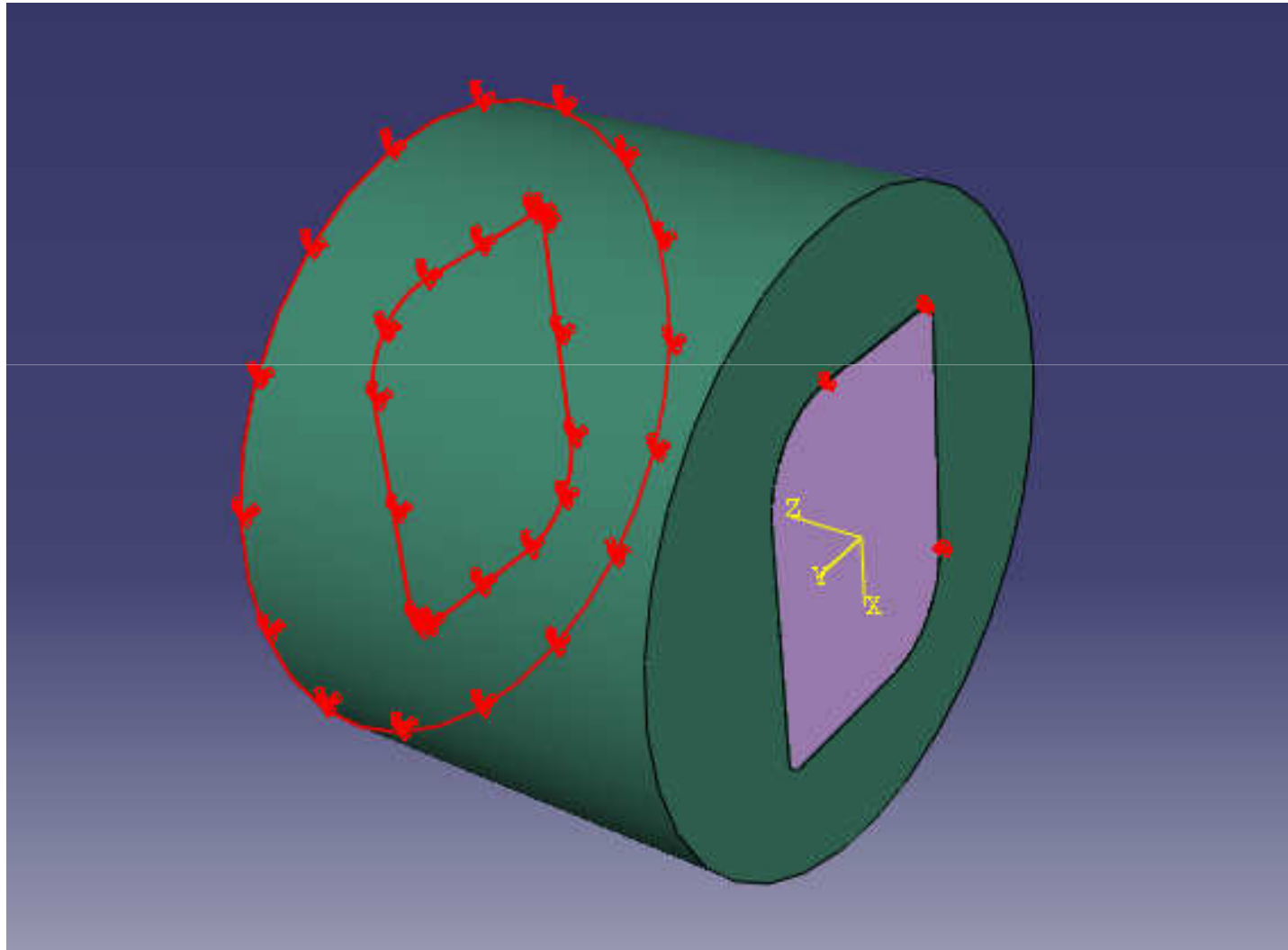
OK Cancel



Element type in thermal and mechanical analysis



Boundary conditions





Output request – nodal temperature

The screenshot displays the ANSYS Workbench interface. On the left, the 'Model Database' tree shows the following structure:

- Model-3_mech
 - Parts (2)
 - Materials (2)
 - Sections (2)
 - Profiles
 - Assembly
 - Steps (2)
 - Initial
 - Step-1
 - Field Output Requests (1)
 - F-Output-1 (Created)
 - History Output Requests (1)
 - ALE Adaptive Mesh Constraints
 - Interactions (1)
 - Loads
 - BCs (4)
 - Predefined Fields (1)
 - Predefined Field-1 (Modified)
 - Load Cases
- Field Output Requests (1)
- History Output Requests (1)

Edit Field Output Request

Name: F-Output-1
Step: Step-1
Procedure: Static, General

Domain: Whole model

Frequency: Every n increments n: 1

Timing: Output at exact times

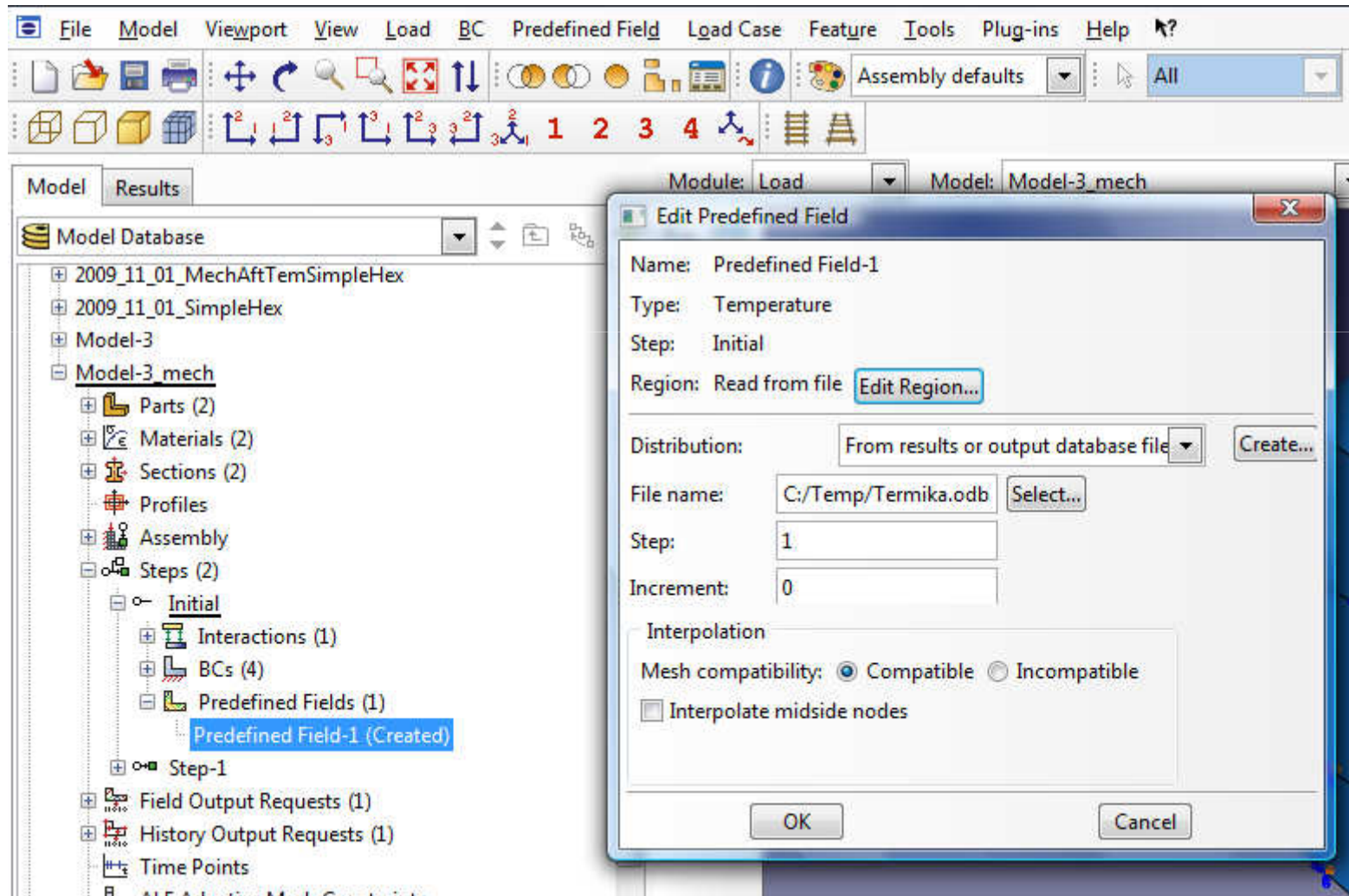
Output Variables

Select from list below Preselected defaults All Edit variables

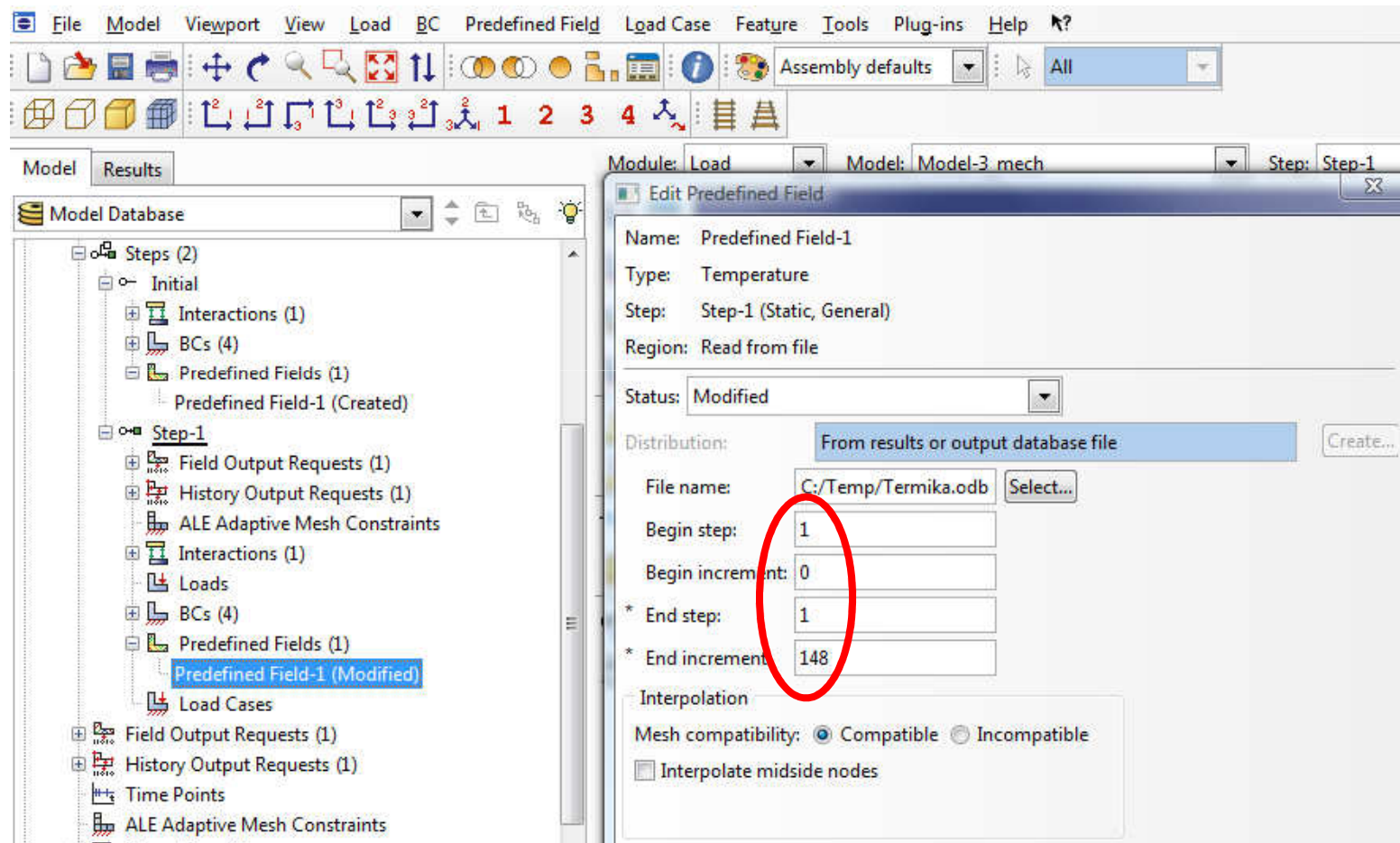
CDISP,CF,CSTRESS,LE,NT,PE,PEEQ,PEMAG,RF,S,U,

- Stresses
- Strains
- Displacement/Velocity/Acceleration
- Forces/Reactions
- Contact
- Energy
- Failure/Fracture
- Thermal
 - NT, Nodal temperature
 - TEMP, Element temperature
 - SJD, Heat flux due to electrical current
 - SJDA, SJD multiplied by the area
 - SJDT, Time integrated SJD

Reading temperature field to mechanical analysis



Definition of increment



Definition of steps and increment which are read from thermal analysis to mechanical analysis (in thermal analysis was 148 increments (monitor))