

Teplotní analýza

ustálené (stacionární) vedení tepla
neustálené (nestacionární) vedení tepla

Ing. Petr Lehner

Co se dozvíme a naučíme?

Analýza vedení tepla – stacionární a nestacionární.

Nastavení materiálových vlastností pro analýzu.

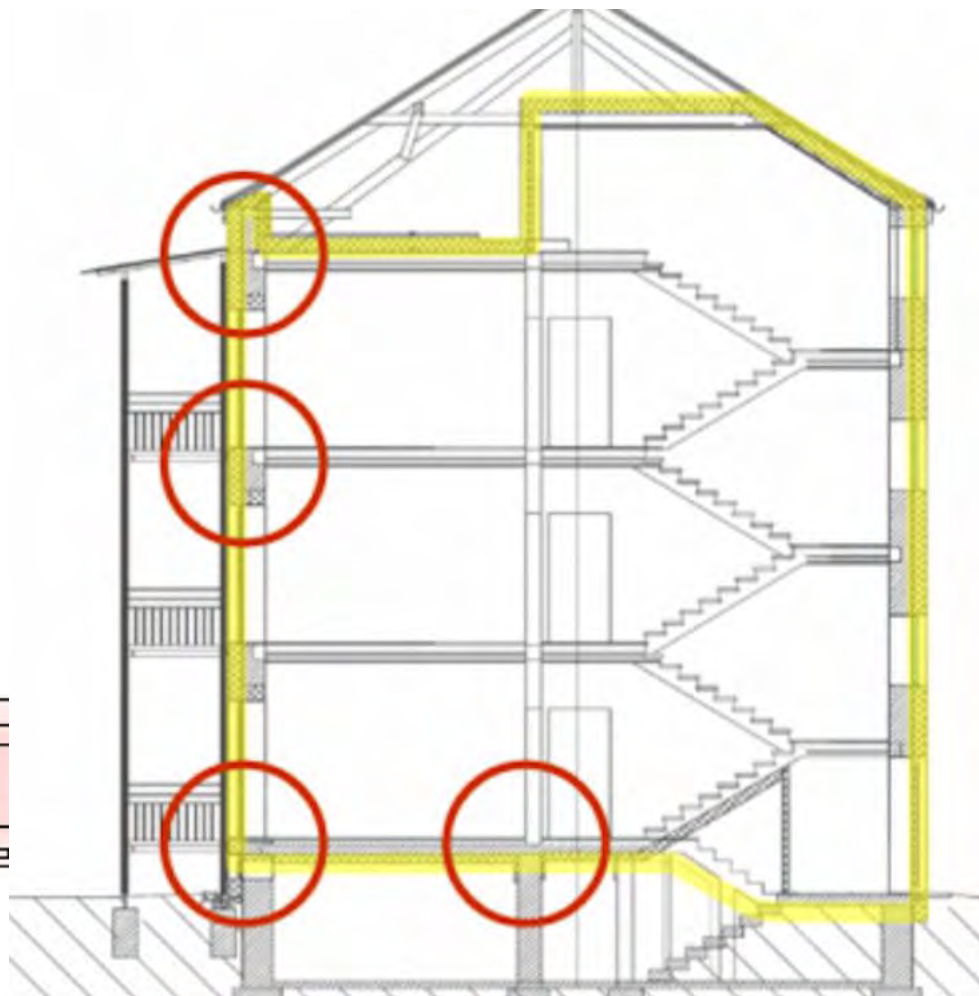
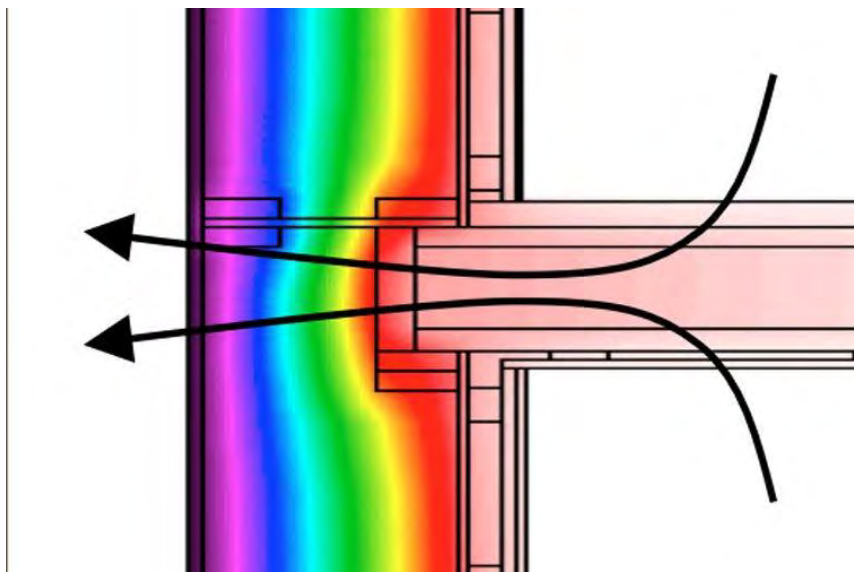
Příprava geometrie z více separovaných částí.

Provázání rozdílných úloh.

Rozdíl mezi stacionárním a nestacionárním vedením.

Tepelný most

- místo v konstrukci, kde konstrukcí domu uniká více tepla než v ostatních místech tepelné obálky objektu.



1) Steady State Thermal – Stacionární analýza

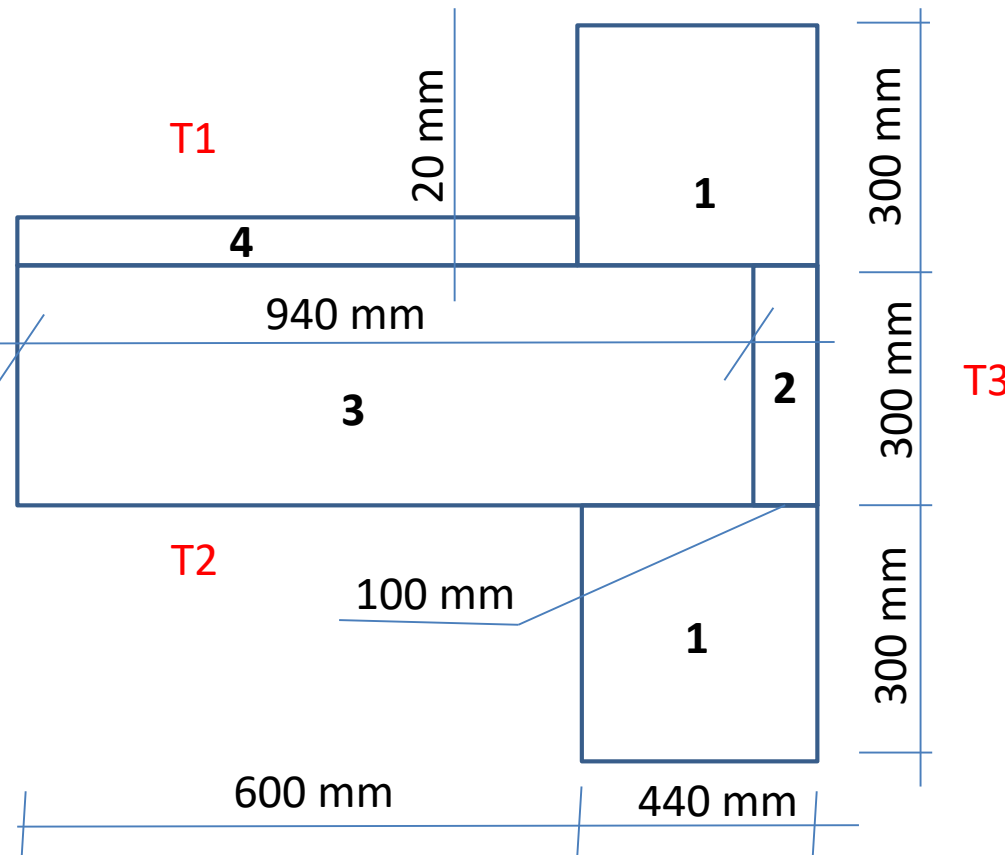
Konstrukce podlahy a stěny

λ = izotropní tepelná vodivost

pozor na jednotky

Tloušťka konstrukce 100 mm

$T_1 = 15\text{ °C}$, $T_2 = 27\text{ °C}$, $T_3 = 10\text{ °C}$



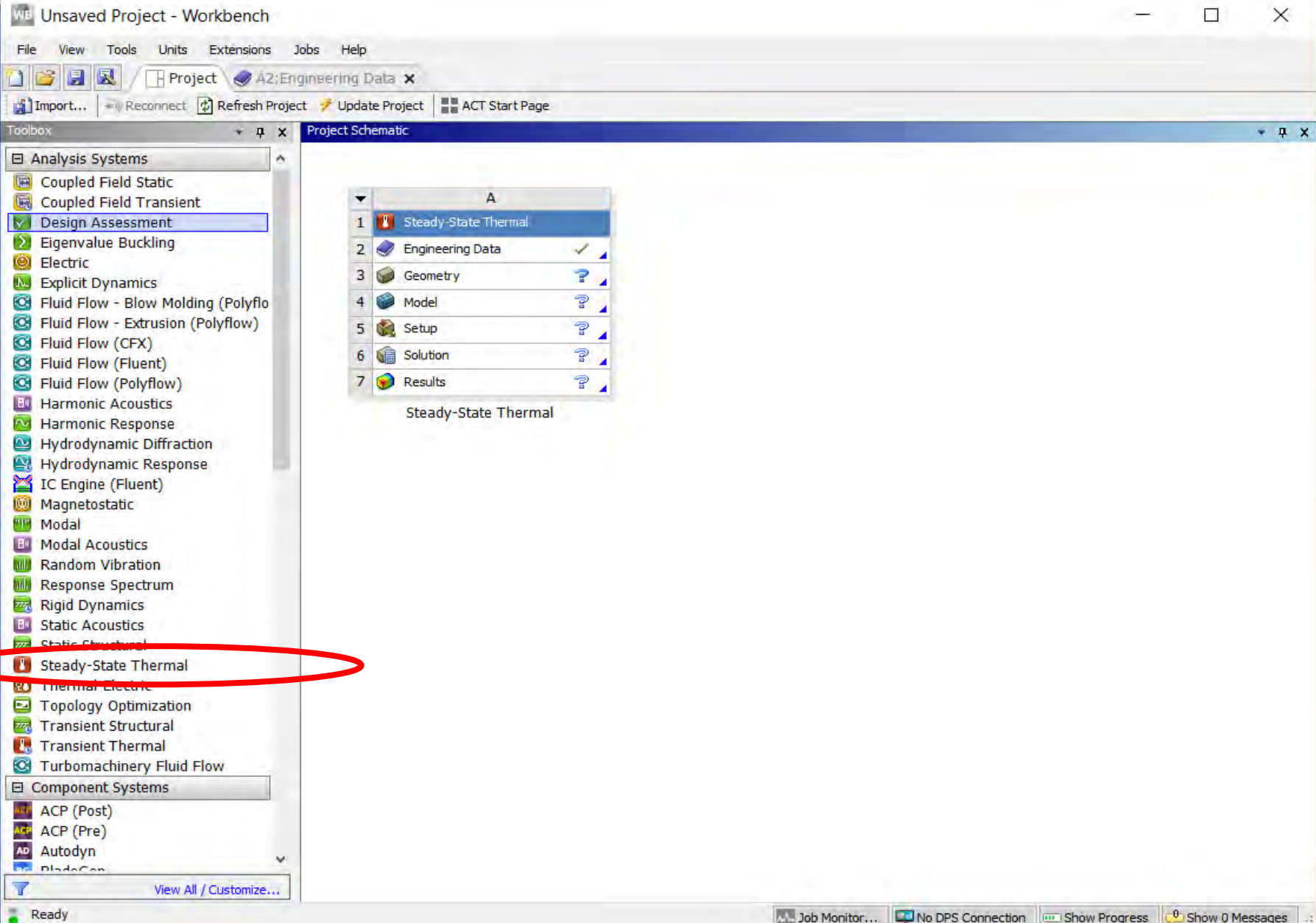
1 – zdivo Porotherm 44
 $\lambda_1 = 0,111\text{ W}\cdot\text{m}^{-1}\cdot\text{K}^{-1}$

2 – tepelná izolace, polystyren
 $\lambda_1 = 0,034\text{ W}\cdot\text{m}^{-1}\cdot\text{K}^{-1}$

3 – železobetonový strop
 $\lambda_1 = 1,58\text{ W}\cdot\text{m}^{-1}\cdot\text{K}^{-1}$

4 – podlaha - korek
 $\lambda_1 = 0,064\text{ W}\cdot\text{m}^{-1}\cdot\text{K}^{-1}$

Stady State Thermal



Engineering data – vlastnosti materiálu

The screenshot shows the ANSYS Workbench Engineering Data environment. The interface includes a menu bar, a toolbar, and several panes. The 'Toolbox' on the left has the 'Thermal' category circled in red. The 'Outline of Schematic A2: Engineering Data' pane shows a tree view of materials, with '04_floor_cork' selected. The 'Table of Properties Row 3: Isotropic Thermal Conductivity' pane shows a table with the value 0,064. The 'Properties of Outline Row 6: 04_floor_cork' pane shows the 'Isotropic Thermal Conductivity' property with a value of 0,064. The 'Chart of Properties Row 3: Isotropic Thermal Conductivity' pane shows a graph of Thermal Conductivity vs. Temperature with a single data point at (0, 0,064). Red callouts point to the 'Thermal' category, the '04_floor_cork' material name, and the '0,064' value.

Annotations:

- Název** (Name): Points to the material name '04_floor_cork' in the Outline of Schematic.
- Hodnota** (Value): Points to the value '0,064' in the Properties of Outline Row 6 table.

	A	B
1	Temperature (C)	Thermal Conductivity (W m ⁻¹ C)
2		0,064
*		

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Isotropic Thermal Conductivity	0,064	W m ⁻¹ ...		

Temperature [C]	Thermal Conductivity [W m ⁻¹ C ⁻¹]
0	0,064

Vytvoření pohledového řezu pomocí obdélníků

The screenshot displays the ANSYS 2020 R1 software interface. The top ribbon includes tabs for File, Design, Display, Assembly, Measure, Facets, Repair, Prepare, Workbench, Detail, Sheet Metal, Tools, and KeyShot. The Design tab is active, and the Measure tool is highlighted with a red circle. The main workspace shows a 3D grid with a cross-section defined by two rectangles. The left sidebar contains the Structure tree (Design1*, Curves), Options - Sketch (Rectangle, Sketch), and Properties. The bottom status bar shows the current coordinates: x=-156,000 y=541,000.

ANSYS 2020 R1

Snapping to Grid

Structure

- Design1*
- Curves

Options - Sketch

- Rectangle
- Define rectangle from center
- Sketch
- Snap to grid
- Snap to angle
- Create layout curves

Properties

Snapping to Grid

x=-156,000 y=541,000

Pull – vytažení ploch do objemu - postupně

The screenshot displays the ANSYS Workbench interface for a Steady-State Thermal analysis. The 'Workbench' tab is highlighted with a red circle. The 'Options - Pull' panel is open, showing the 'General' section with the 'No merge' option selected. A red callout box points to this option with the text: „no marge“ aby nevznikl jeden celek. The 3D model shows a green rectangular block being pulled into an orange rectangular block. The 'Structure' tree on the left shows 'Design1*' with 'Surface' and 'Solid' checked. The 'Properties' panel shows 'Appearance' and 'Face Type' settings.

ANSYS 2020 R1

„no marge“ aby nevznikl jeden celek

Přejmenovat a uzamknout - Rename

The screenshot displays the ANSYS 2020 R1 software interface. The main window shows a 3D model of a mechanical part with a callout box pointing to the 'Design1*' folder in the Structure tree. The callout box contains the text: „rename“ pro každou část po vytažení. The Structure tree on the left shows the following items: Design1*, Surface, wall up, wall down, and a folder icon. The Properties panel at the bottom left shows the Appearance and Material properties for the selected part. The bottom status bar indicates 'Pull 6 faces' and '1 Body'.

ANSYS
2020 R1

„rename“ pro každou část po vytažení

Structure

- Design1*
- Surface
- wall up
- wall down
- Folder icon

Options - Pull

General

+ Add - Cut No merge

Properties

Appearance

Color ARGB: 255, 143, ...

Style By Layer, By Style

Tessellation Que: 5

Material

Material Name Unknown Material

Fluid False

Pull 6 faces

1 Body

Kontrola, zda jsou všechny komponenty odděleny

The screenshot displays the ANSYS 2020 R1 software interface. The main window shows a 3D model of a cross-shaped structure. The top menu bar includes File, Design, Display, Assembly, Measure, Facets, Repair, Prepare, Workbench, Detail, Sheet Metal, Tools, and KeyShot. The ribbon contains various tool categories: Home, Spin, Plan View, Part, Zoom, Orient, Sketch, Mode, Edit, Intersect, Create, and Body. The Structure tree on the left is highlighted with a red circle and lists the following components: Design 1*, wall up, wall down, T1, ceil, and Solid. The Properties panel at the bottom left shows the Appearance and Face Type settings. The bottom status bar indicates '1 Face' is selected.

Click an object. Double-click to select an edge loop. Triple-click to select a solid.

ANSYS 2020 R1

Structure

- Design 1*
- wall up
- wall down
- T1
- ceil
- Solid

Appearance

Color: ARGB: 255, 143, 175

Style: By Layer, By Style

Face Type

Shape: Plane

Chamfer: False

1 Face

Tvorba důležitých hran na plochách- split

The screenshot shows the ANSYS 2020 R1 software interface. The ribbon is set to the 'Design' tab, and the 'Split' tool is highlighted in the 'Intersect' group. A callout box points to the 'Split' tool with the following text:

označit plochu
> split
> najít na **hraně**
plochy bod dle
sousední plochy
nebo hrany
(lepší tvorba
sítě)

Below the callout, a 3D model of a cross-shaped part is shown. The 'Split' tool is being used to split a face of the model. A callout box at the bottom of the image contains the following text:

pozor – tento model obsahuje 5 takových hran

The interface also shows the 'Structure' tree on the left, the 'Properties' panel at the bottom left, and the 'Click a point on an edge to split from' prompt above the model.

Model – geometry – přiřazení materiálů

The screenshot displays the ANSYS 2020 R1 Academic software interface. The main window shows a 3D model of a thermal assembly with a green top layer, a brown middle layer, and a blue bottom layer. The interface is divided into several panels:

- Outline:** Shows the project hierarchy. The **Model (A4)** folder is circled in red, containing sub-entities like **SYS\wall up**, **SYS\wall down**, **SYS\TI**, **SYS\ceil**, and **SYS\cork**.
- Details of "SYS\wall up":** Shows the material assignment for the selected entity. The **Assignment** field is circled in red, showing a yellow material icon.
- Engineering Data Materials:** A list of materials is shown, with **01_porotherm** circled in red. Other materials include **Water Liquid**, **Structural Steel**, **Air**, **04_floor_cork**, **03_reinforced ceiling**, and **02_thermal insulation**.
- Messages:** A panel at the bottom showing the current message: **Association**.

The status bar at the bottom indicates the software is **Ready** and shows units: **Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius**.

Mech – element size 25 mm - velikost

The screenshot displays the ANSYS Academic Research Mechanical and CFD software interface. The main window shows a 3D model of a cross-shaped part with a mesh. The 'Details of Mesh' panel is open, showing the following settings:

Details of "Mesh"	
Display Style	Use Geometry Setting
Physics Preference	Mechanical
Element Order	Program Controlled
Element Size	25, mm
Sizing	
Quality	
Inflation	
Advanced	
Statistics	
Nodes	20755
Elements	3912

The 'Outline' panel on the left shows the project structure, with the 'Mesh' object selected and circled in red. The 'Messages' panel at the bottom is empty.

ANSYS 2020 R1 ACADEMIC

0,00 200,00 400,00 (mm)

Ready No Messages No Selection Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius

Temperatures – na plochy = 10 °C

ANSYS 2020 R1 ACADEMIC

A: Steady-State Thermal
Temperature
Time: 1, s
14.03.2021 12:24

Temperature: 10, °C

Details of "Temperature"

Scope	
Scoping Method	Geometry Selection
Geometry	3 Faces
Definition	
Type	Temperature
<input type="checkbox"/> Magnitude	10, °C (ramped)
<input checked="" type="checkbox"/> Suppressed	No

Graph

22, 10, 1, 1

Tabular Data

Steps	Time [s]	Temperature [°C]
1	0,	= 22,
2	1,	10,
*		

0,00 200,00 400,00 (mm)

Messages Graph

No Messages No Selection Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius

Temperatures – na plochy = 27 °C

Context
A : Steady-State Thermal - Mechanical [ANSYS Academic Research Mechanical and CFD]

File Home Environment Display Selection Automation Quick Launch

Isometric Previous Rotate +Sx Rotate -Sx Pan Up Pan Down
Look At Next Rotate +Sy Rotate -Sy Pan Left Pan Right
Views Angle 10 Rotate +Sz Rotate -Sz Zoom In Zoom Out
Orient

Random Rescale Display Show Mesh Thick Shells and Beams Cross Section Display Style Vertex Edge Explode Viewports Show Display

Outline Details of "Temperature"

Project*
Model (A4)
Geometry
SYS\wall up
SYS\wall down
SYS\TI
SYS\ceil
SYS\cork
Materials
04_floor_cork
03_reinforced ceiling
02_thermal insulation
01_porotherm
Coordinate Systems
Connections
Mesh
Steady-State Thermal (A)
Initial Temperature
Analysis Settings
Temperature
Solution (A6)
Solution Information

Scope
Scoping Method: Geometry Selection
Geometry: 3 Faces
Definition
Type: Temperature
Magnitude: 27, °C (ramped)
Suppressed: No

A: Steady-State Thermal
Temperature
Time: 1, s
14.03.2021 12:26
Temperature: 27, °C

ANSYS 2020 R1 ACADEMIC

0,00 400,00 (mm) 200,00

Graph
27, 22, 1, 1

Tabular Data

Steps	Time [s]	Temperature [°C]
1	0,	= 22,
2	1,	27,
*		

Ready No Messages 2 Faces Selected: Angle = 90.° Area = 90000 mm² Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius

Temperatures – na plochy = 15 °C

ANSYS 2020 R1 ACADEMIC

A: Steady-State Thermal - Mechanical [ANSYS Academic Research Mechanical and CFD]

Temperature 3
Time: 1, s
14.03.2021 12:27

Temperature 3: 15, °C

0,00 200,00 400,00 (mm)

Outline

- Project*
 - Model (A4)
 - Geometry
 - SYS\wall up
 - SYS\wall down
 - SYS\TI
 - SYS\ceil
 - SYS\cork
 - Materials
 - 04_floor_cork
 - 03_reinforced ceiling
 - 02_thermal insulation
 - 01_porotherm
 - Coordinate Systems
 - Connections
 - Mesh
 - Steady-State Thermal (A)
 - Initial Temperature
 - Analysis Settings
 - Temperature
 - Temperature 2
 - Temperature 3
 - Solution (A6)
 - Solution Informati

Details of "Temperature 3"

Scope	
Scoping Method	Geometry Selection
Geometry	2 Faces
Definition	
Type	Temperature
Magnitude	15, °C (ramped)
Suppressed	No

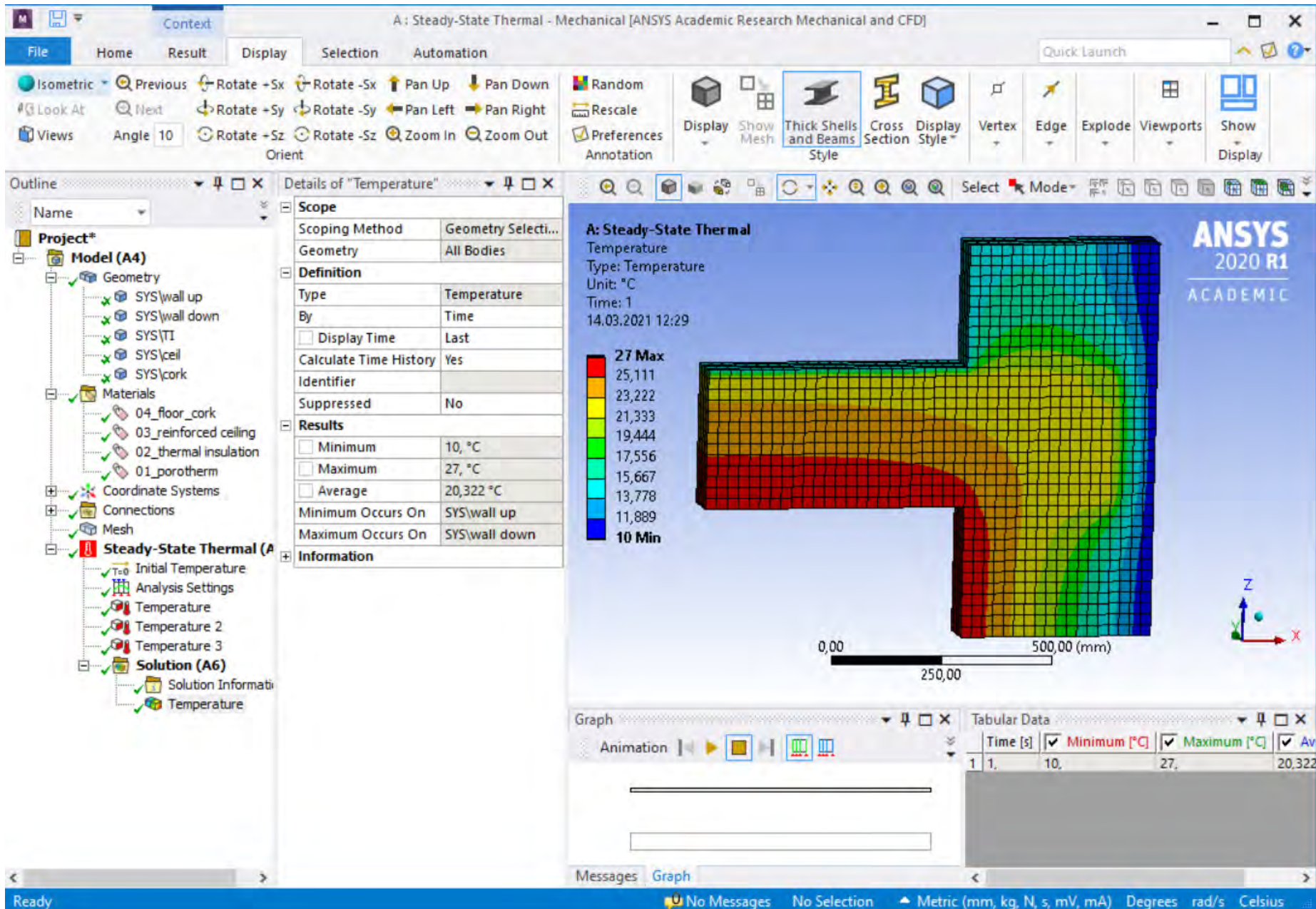
Graph

Tabular Data

Steps	Time [s]	Temperature [°C]
1	0,	= 22,
2	1,	15,
*		

Ready | No Messages | No Selection | Metric (mm, kg, N, s, mV, mA) | Degrees | rad/s | Celsius

Temperature - výsledky



1) Steady State Thermal – Stacionární analýza

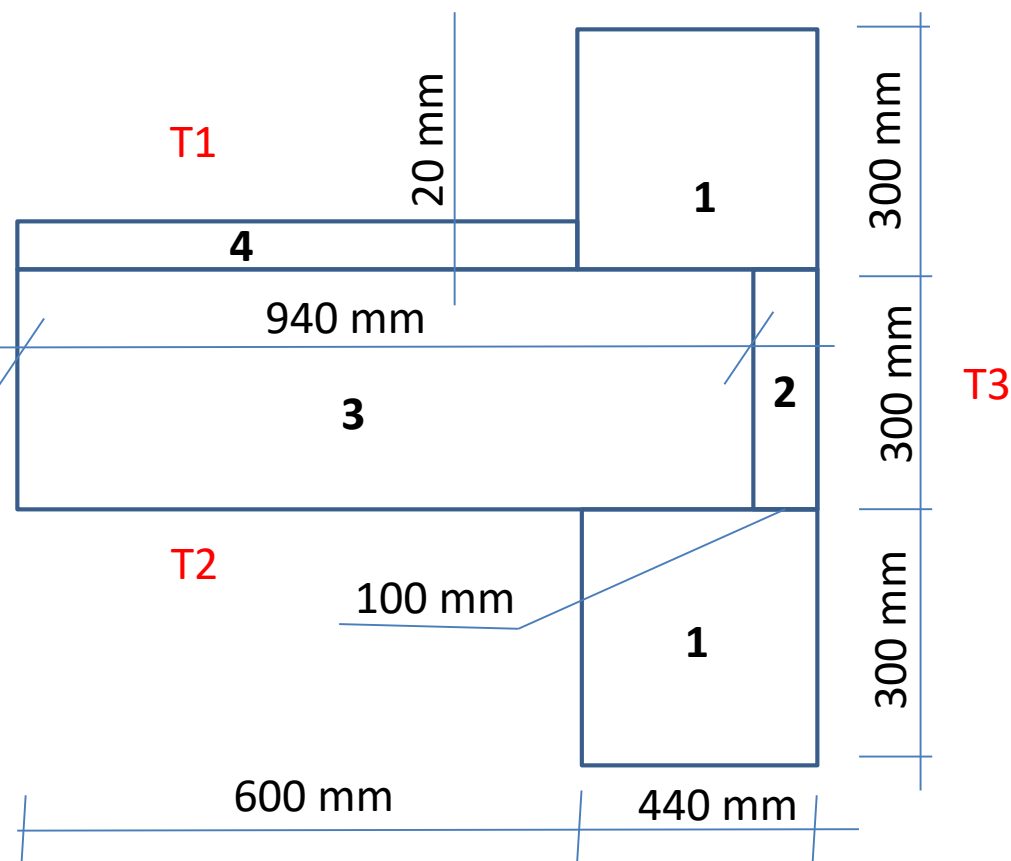
Konstrukce podlahy a stěny

λ = izotropní tepelná vodivost

pozor na jednotky

Ploušťka konstrukce 100 mm

$T_1 = 15\text{ °C}$, $T_2 = 27\text{ °C}$, $T_3 = 10\text{ °C}$



1 – zdivo Porotherm 44
 $\lambda_1 = 0,111\text{ W}\cdot\text{m}^{-1}\cdot\text{K}^{-1}$

2 – tepelná izolace, polystyren
 $\lambda_1 = 0,034\text{ W}\cdot\text{m}^{-1}\cdot\text{K}^{-1}$

3 – železobetonový strop
 $\lambda_1 = 1,58\text{ W}\cdot\text{m}^{-1}\cdot\text{K}^{-1}$

4 – podlaha - korek
 $\lambda_1 = 0,064\text{ W}\cdot\text{m}^{-1}\cdot\text{K}^{-1}$

2) Transient Thermal - Nestacionární analýza

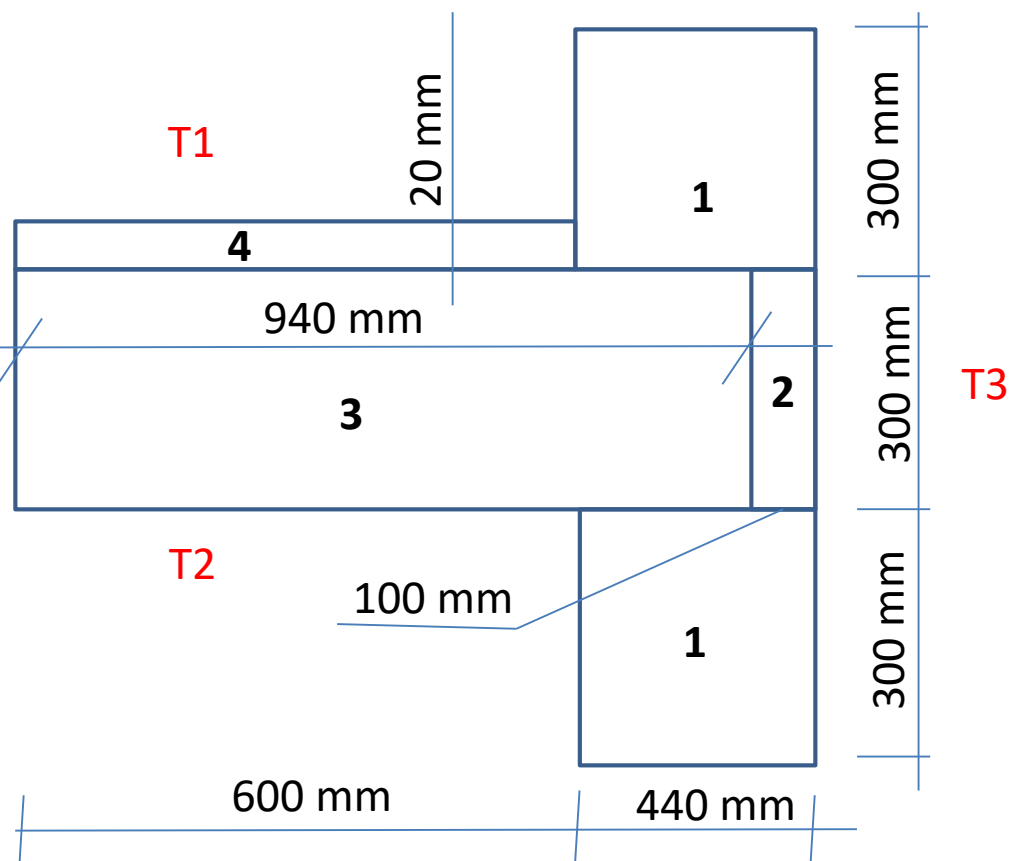
Konstrukce podlahy a stěny

λ = izotropní tepelná vodivost

c = měrná tepelná kapacita

ρ = hustota

pozor na jednotky



Ploušťka konstrukce 100 mm

$T1 = 15\text{ °C}$, $T2 = 27\text{ °C}$, $T3 = 10\text{ °C}$

1 – zdivo Porotherm 44

$\lambda_1 = 0,111\text{ W}\cdot\text{m}^{-1}\cdot\text{C}^{-1}$

$c_1 = 960\text{ J}\cdot\text{kg}^{-1}\text{ C}^{-1}$,

$\rho_1 = 750\text{ kg/m}^3$

2 – tepelná izolace, polystyren

$\lambda_1 = 0,034\text{ W}\cdot\text{m}^{-1}\cdot\text{C}^{-1}$

$c_1 = 2060\text{ J}\cdot\text{kg}^{-1}\text{ C}^{-1}$,

$\rho_1 = 30\text{ kg/m}^3$

3 – železobetonový strop

$\lambda_1 = 1,58\text{ W}\cdot\text{m}^{-1}\cdot\text{C}^{-1}$

$c_1 = 1020\text{ J}\cdot\text{kg}^{-1}\text{ C}^{-1}$,

$\rho_1 = 2400\text{ kg/m}^3$

4 – podlaha - korek

$\lambda_1 = 0,064\text{ W}\cdot\text{m}^{-1}\cdot\text{C}^{-1}$

$c_1 = 1880\text{ J}\cdot\text{kg}^{-1}\text{ C}^{-1}$,

$\rho_1 = 150\text{ kg/m}^3$

Transient Thermal – kopie materiálu a geometrie

The screenshot shows the ANSYS Workbench interface. The Project Schematic displays two analysis systems, A and B. System A is a Steady-State Thermal analysis, and System B is a Transient Thermal analysis. A blue line connects the 'Geometry' (row 3) of System A to the 'Geometry' (row 3) of System B. The 'Engineering Data' (row 2) of System B has a question mark icon, indicating it is not yet defined. The 'Transient Thermal' option in the Toolbox is circled in red. A red callout box contains the following text:

nová úloha
> propojit engineering data
a geometry
> vymazat propojení u ED

Engineering data – nastavení nových parametrů

The screenshot displays the ANSYS Workbench Engineering Data environment. The interface is divided into several panes:

- Toolbox:** A list of material properties and models. Two items are circled in red: "Physical Properties" and "Specific Heat, C_p ".
- Outline of Schematic B2: Engineering Data:** A table listing materials and their properties. Row 3, "01_porotherm", is selected.
- Properties of Outline Row 3: 01_porotherm:** A table showing the material's properties. Row 5, "Specific Heat, C_p ", is circled in red.
- Table of Properties Row 5: Specific Heat, C_p :** A table showing the specific heat value for the selected material. Row 2 shows a value of 960.
- Chart of Properties Row 5: Specific Heat, C_p :** A graph showing the specific heat value (Y-axis, $10^3 \text{ J kg}^{-1} \text{ C}^{-1}$) versus temperature (X-axis, C). A single data point is plotted at $(0, 0.96)$.

The status bar at the bottom indicates "Ready" and shows system information like "Job Monitor...", "No DPS Connection", "Show Progress", and "Show 0 Messages".

Model – geometry – přiřazení materiálů

The screenshot displays the ANSYS 2020 R1 Academic software interface. The main window shows a 3D model of a wall assembly with various layers. The interface is divided into several panes:

- Outline:** Shows the project hierarchy. The "Model (B4)" folder is expanded, and its sub-items are circled in red: "SYS\wall up", "SYS\wall down", "SYS\TI", "SYS\ceil", and "SYS\cork".
- Details of "SYS\wall down":** Shows the properties of the selected part. The "Material" section is expanded, and the "Assignment" row is circled in red. The "Assignment" row shows a yellow material assigned to the part.
- Engineering Data Materials:** A list of materials is shown, with "01_porotherm" circled in red. Other materials listed include "Water Liquid", "Structural Steel", "Air", "04_floor_cork", "03_reinforced ceiling", and "02_thermal insulation".
- 3D Model:** Shows a 3D view of the wall assembly with a scale bar from 0,00 to 500,00 (mm). The model is composed of several layers: a top grey layer, a brown layer, a green layer, and a bottom green layer.
- Messages:** Shows the "Text" and "Association" panes.

The status bar at the bottom indicates "Ready" and "Messages pane No Selection". The units are set to "Metric (mm, kg, N, s, mV, mA)", and the temperature unit is "Degrees".

Mech – element size 25 mm – velikost sítě

The screenshot displays the ANSYS 2020 R1 Academic software interface. The main window shows a 3D model of a cross-shaped part with a mesh. The 'Details of Mesh' panel is open, showing the following settings:

Details of "Mesh"	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
Element Size	25, mm
Sizing	
Quality	
Inflation	
Advanced	
Statistics	
Nodes	20755
Elements	3912

The 'Outline' tree on the left shows the project structure, with the 'Mesh' object selected. The 'Messages' panel at the bottom shows the following message:

Text	Association
The default Error Limit for Mechanical Physics Preference has changed to Aggressive Mect	Project>Model>Me

Analysis Settings – 86400 s (24 hodin)

Details of "Analysis Settings"

Step Controls	
Number Of Steps	1,
Current Step Number	1,
Step End Time	86400 s
Auto Time Stepping	Program Controlled
Initial Time Step	864, s
Minimum Time Step	86,4 s
Maximum Time Step	8640, s
Time Integration	On
Solver Controls	
Solver Type	Program Controlled
Radiosity Controls	
Nonlinear Controls	
Advanced	
Output Controls	
Analysis Data Management	
Visibility	

Graph

Tabular Data

Steps	End Time
1	86400
*	

Messages | Graph

1 Message No Selection Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius

Temperatures – na plochy, 15 °C, 27 °C, 10 °C

The screenshot displays the ANSYS Academic Research Mechanical and CFD software interface. The main window shows a 3D model of a mechanical part with temperature results. The software is running a transient thermal analysis. The temperature results are shown as a color map on the model, with a red color indicating a temperature of 15 °C. The software version is ANSYS 2020 R1 Academic.

Details of "Temperature 3"

Scoping Method	Geometry Selection
Geometry	2 Faces
Definition	
Type	Temperature
Magnitude	15, °C (step applied)
Suppressed	No

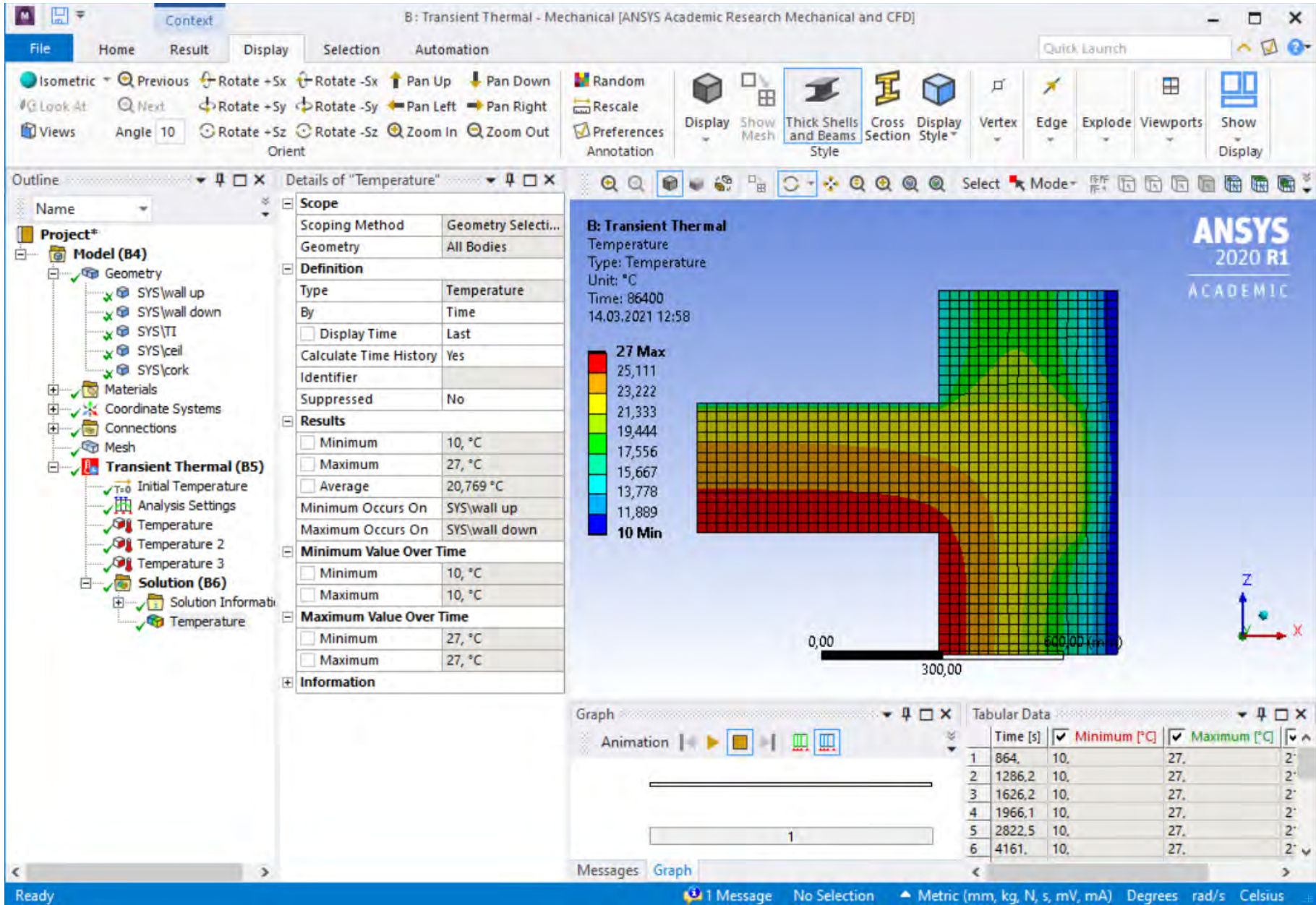
Graph

Steps	Time [s]	Temperature [°C]
1	0,	= 15,
2	86400	15,
*		

Tabular Data

Steps	Time [s]	Temperature [°C]
1	0,	= 15,
2	86400	15,
*		

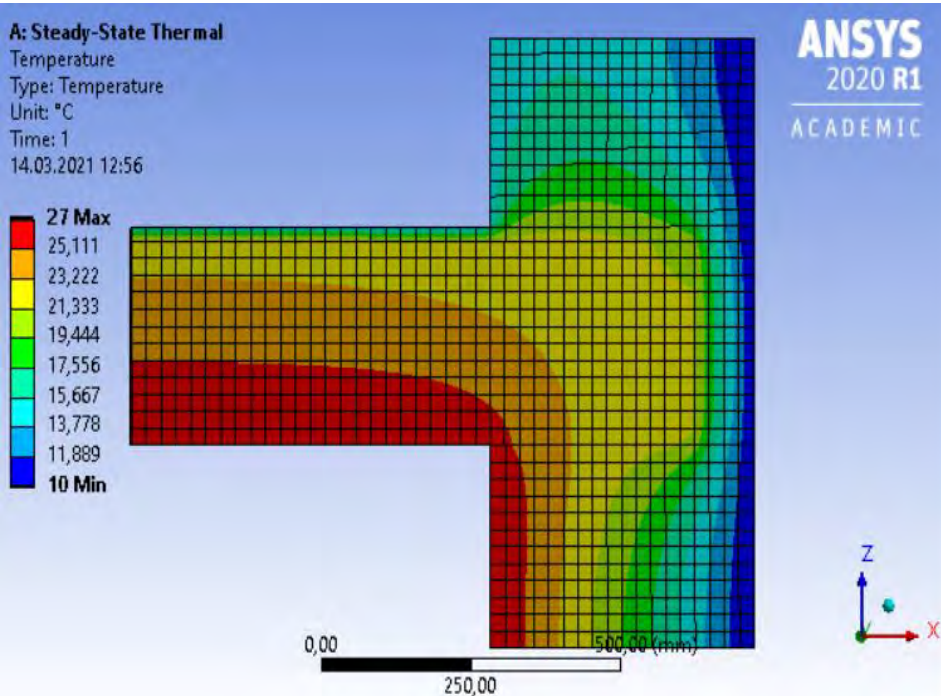
Temperature - výsledky



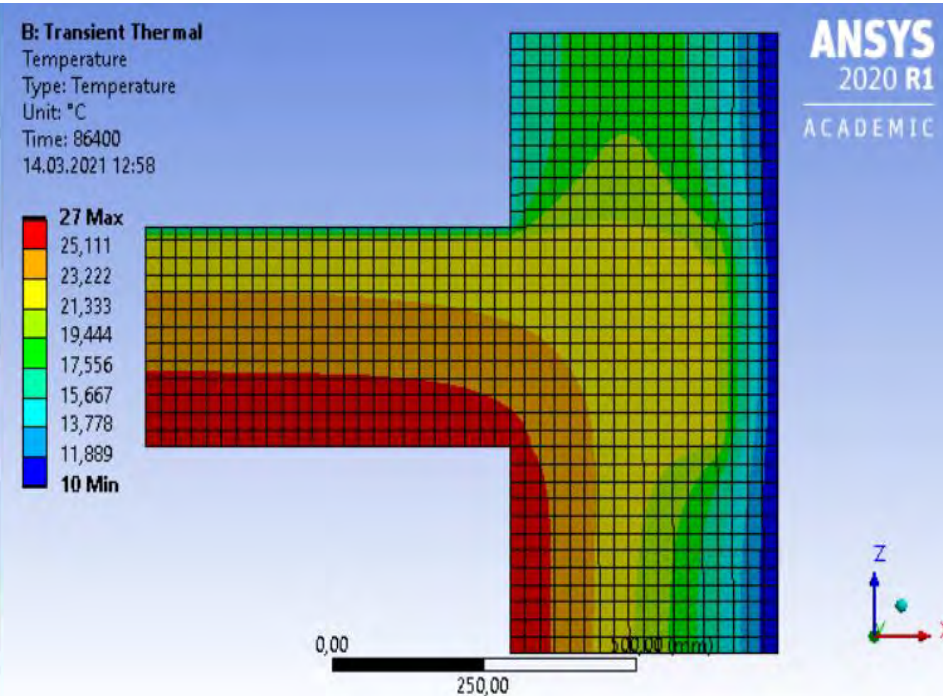
Temperature – porovnání

Stady State

Transient Thermal



ustálený stav



stav po 24 hodinách

Display time – 1800 s (5 hodin) – evaluate results

The image displays the ANSYS 2020 R1 Academic software interface for a transient thermal simulation. The main window shows a 3D model of a mechanical part with a temperature distribution plot. The plot uses a color scale from blue (10 Min) to red (27 Max). The 'Details of Temperature' panel is open, showing the following settings:

- Scope: Scoping Method: Geometry Selecti..., Geometry: All Bodies
- Definition: Type: Temperature, Display Time: 18000 s (circled in red), Calculate Time History: Yes
- Results: Minimum: 10, °C, Maximum: 27, °C, Average: 21,139 °C
- Minimum Value Over Time: Minimum: 10, °C, Maximum: 10, °C
- Maximum Value Over Time: Minimum: 27, °C, Maximum: 27, °C

The 'Graph' panel at the bottom shows a table of results:

Time [s]	Minimum [°C]	Maximum [°C]
1	864	10
2	1286.2	10
3	1626.2	10
4	1966.1	10
5	2822.5	10
6	4161	10

The status bar at the bottom indicates 'Ready', '1 Message', 'No Selection', and 'Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius'.