

Sdružená úloha – teplo, statické zatížení

Ing. Petr Lehner

Co se dozvíme a naučíme?

Kombinace různých typů úloh.

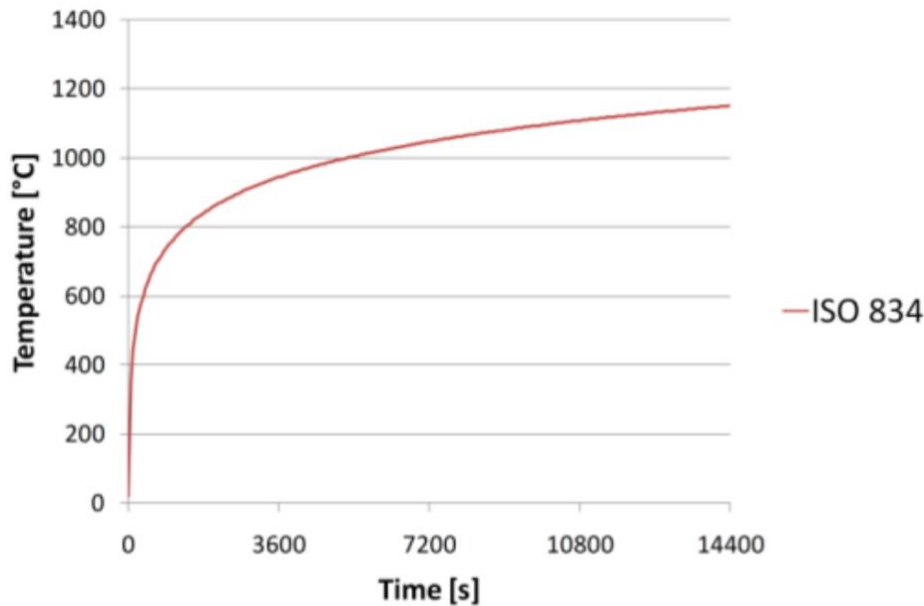
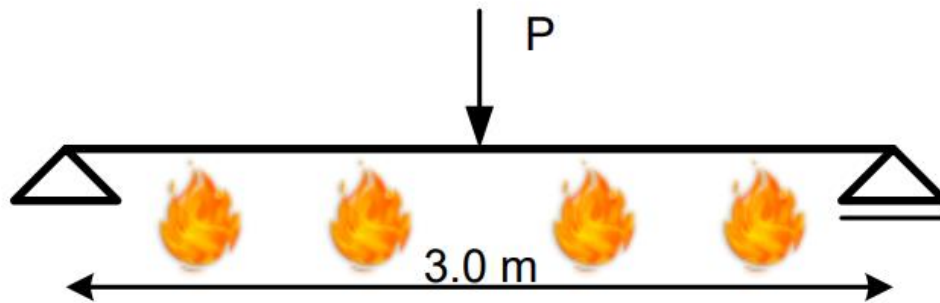
Přenos vlastností a výsledků mezi modely.

Zadávání zatížení pomocí tabulkových hodnot.

Vyhodnocení časově závislého procesu.

Porovnání variant.

Nosník zatížený silou a teplotou od požáru



Důvod

- chování samostatně a při sdružené úloze

Materiál

- ocel (default)

Konstrukce

- Space claim

Varianty

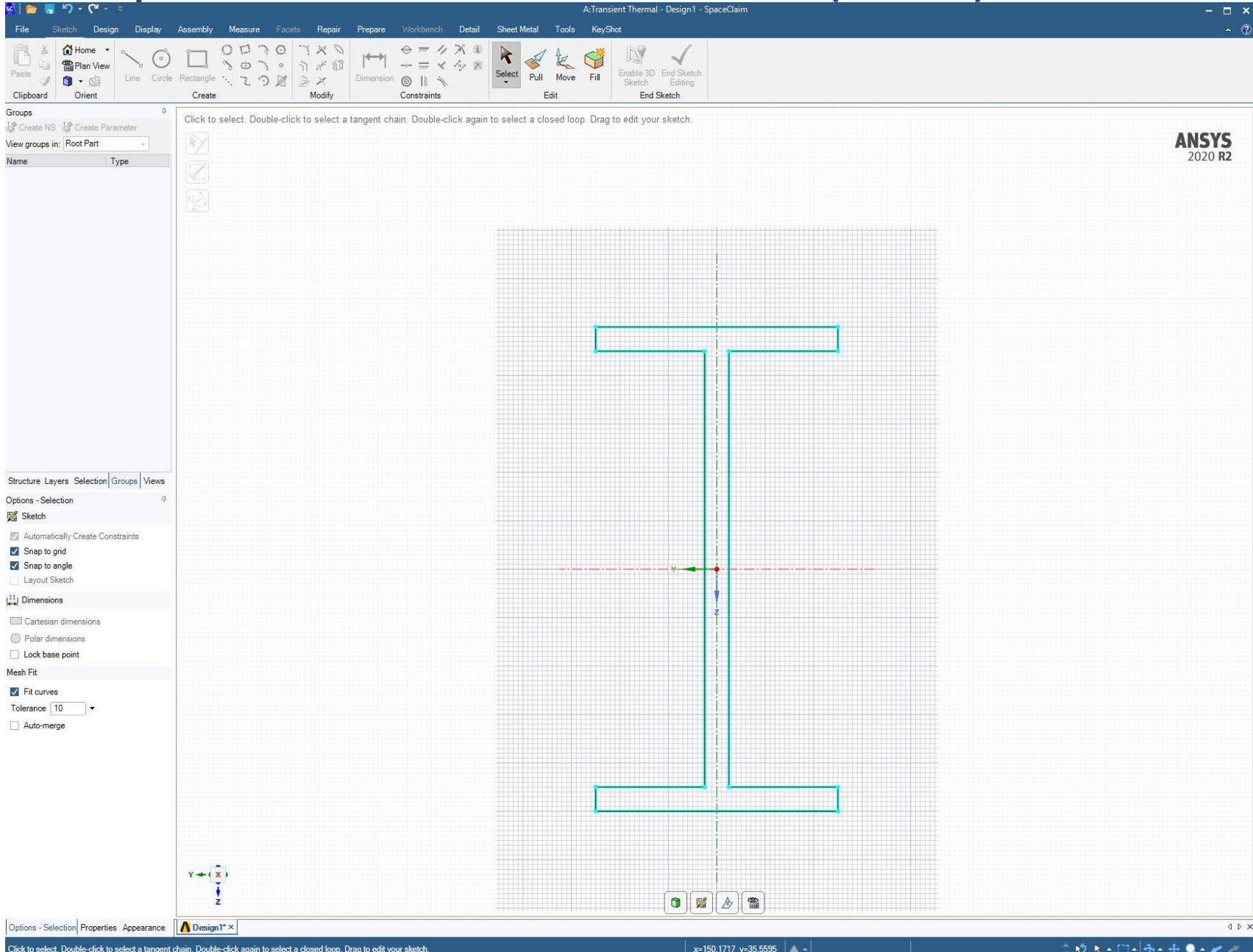
- průběh teplot samostatně
- zatížení samostatně
- společné působení

Transient Thermal + Static Structural

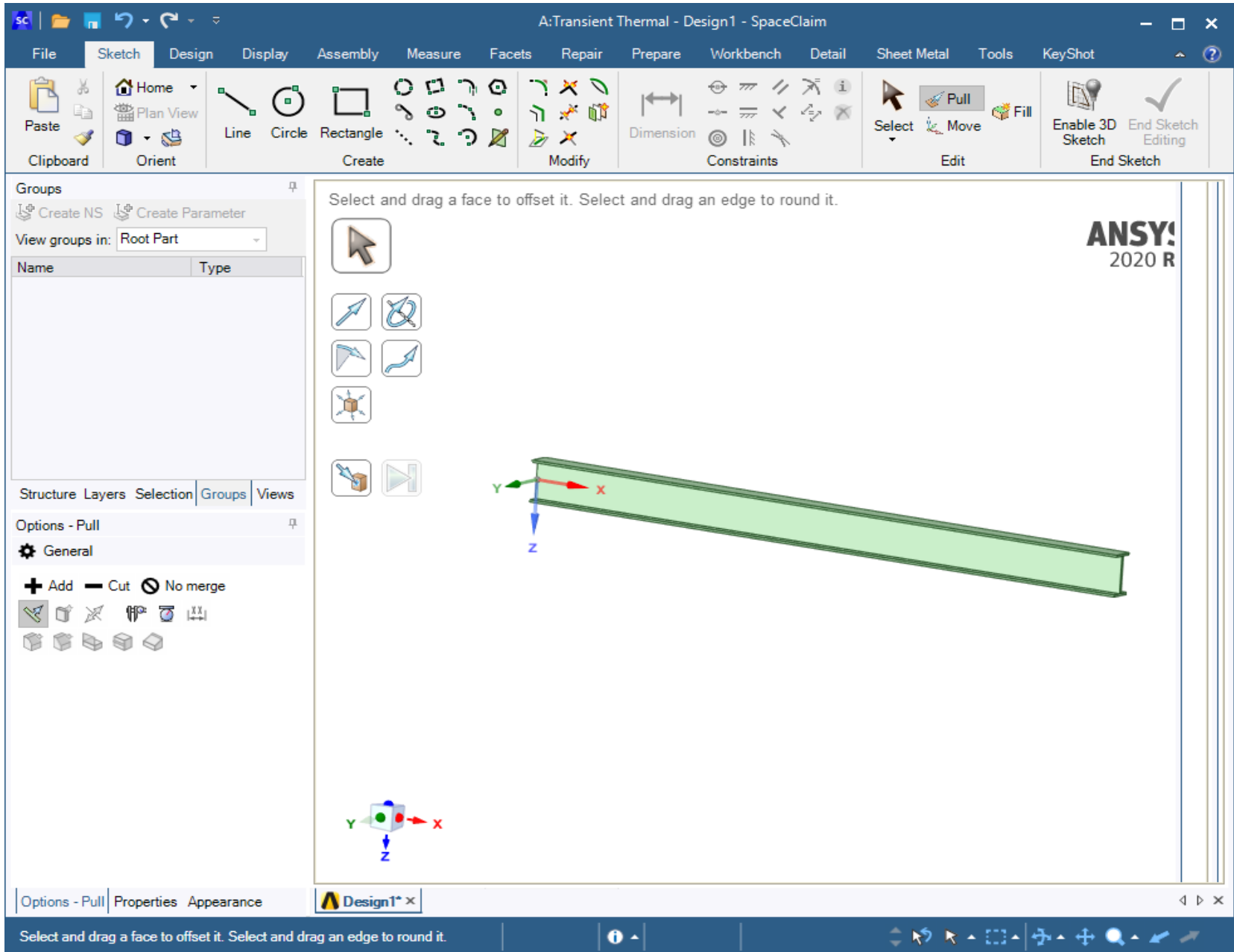
The screenshot displays the ANSYS Workbench interface. The Project Schematic shows two analysis systems, A and B, connected by a red arrow. System A is 'Transient Thermal' and System B is 'Static Structural'. A red callout box with the word 'Propojení' (Connection) points to the arrow. The Properties of Project Schematic panel on the right shows a table with columns A and B.

	A	B
1	Property	Value
2	Notes	
3	Notes	
4	Solution Process	
5	Update Option	Run in Foreground

Space Claim – řez 200 x 100 (tl. 10) mm



Space Claim – PULL – délka 3000 mm



Model – Mesh - default

The screenshot displays the ANSYS Mechanical Enterprise software interface. The main window title is "A : Transient Thermal - Mechanical [Ansys Mechanical Enterprise]". The software is running in the "Context" mode, with the "Mesh" tab selected in the top toolbar. The toolbar contains various meshing tools such as Duplicate, Solve, Insert, Update, Generate, Surface Mesh, Source/Target, Method, Sizing, Face Meshing, Mapped Meshing, Mesh Copy, Match Control, Contact Sizing, Refinement, Inflation, Gasket, Mesh Group, Mesh Edit, and Metrics Display.

The Outline tree on the left shows the project structure:

- Project*
- Model (A4)
 - Geometry
 - Materials
 - Coordinate Systems
 - Mesh
- Transient Thermal (A5)
 - Initial Temperature
 - Analysis Settings
- Solution (A6)
 - Solution Information
 - Temperature - Global Ma
 - Temperature - Global Min

The Details of "Mesh" panel shows the following settings:

Display	Element Quality
Display Style	Element Quality
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
Element Size	Default
Sizing	
Quality	
Inflation	
Advanced	
Statistics	

The central 3D view area shows a meshed beam. The mesh is color-coded by element quality, with a scale ranging from 0.028903 (Min) to 0.098756 (Max). The ANSYS 2020 R2 logo is visible in the top right corner of the 3D view. A scale bar at the bottom of the 3D view indicates a length of 700.00 mm, with a midpoint at 350.00 mm. The status bar at the bottom shows "Ready" and "Messages pane".

Model – Mesh – 25 mm

The screenshot displays the ANSYS Mechanical Enterprise interface for a transient thermal-mechanical analysis. The main 3D view shows a beam model with a mesh. A color scale for element quality is visible, ranging from 0.48937 (Min) to 0.60407 (Max). The Details panel for the Mesh shows the element size set to 25 mm. A red callout box points to this value.

Element Quality
0.60407 Max
0.59132
0.57858
0.56584
0.55309
0.54035
0.5276
0.51486
0.50212
0.48937 Min

Details of "Mesh"	
Display	Element Quality
Display Style	Element Quality
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
<input type="checkbox"/> Element Size	25, mm
Sizing	
Quality	
Inflation	
Advanced	
Statistics	

25 mm

0,00 300,00 600,00 (mm)

Messages

Text Association

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

Transient Thermal – Analysis Settings - 7200 s

The screenshot displays the ANSYS Mechanical Enterprise interface for a Transient Thermal analysis. The main window shows a 3D model of a meshed beam. The **Details of "Analysis Settings"** panel is open, showing the following settings:

Property	Value
Number Of Steps	1
Current Step Number	1
Step End Time	7200, s
Auto Time Stepping	Program Controlled
Initial Time Step	2, s
Minimum Time Step	7,2 s
Maximum Time Step	720, s
Time Integration	On
Solver Type	Program Controlled
Stiffness Controls	
Nonlinear Controls	
Advanced	
Output Controls	
Analysis Data Management	
Visibility	

A red callout box highlights the value **7200 s** in the Step End Time field.

The **Messages** and **Tabular Data** windows are also visible. The **Tabular Data** window shows the following data:

Steps	End Time [s]
1	7200,
*	

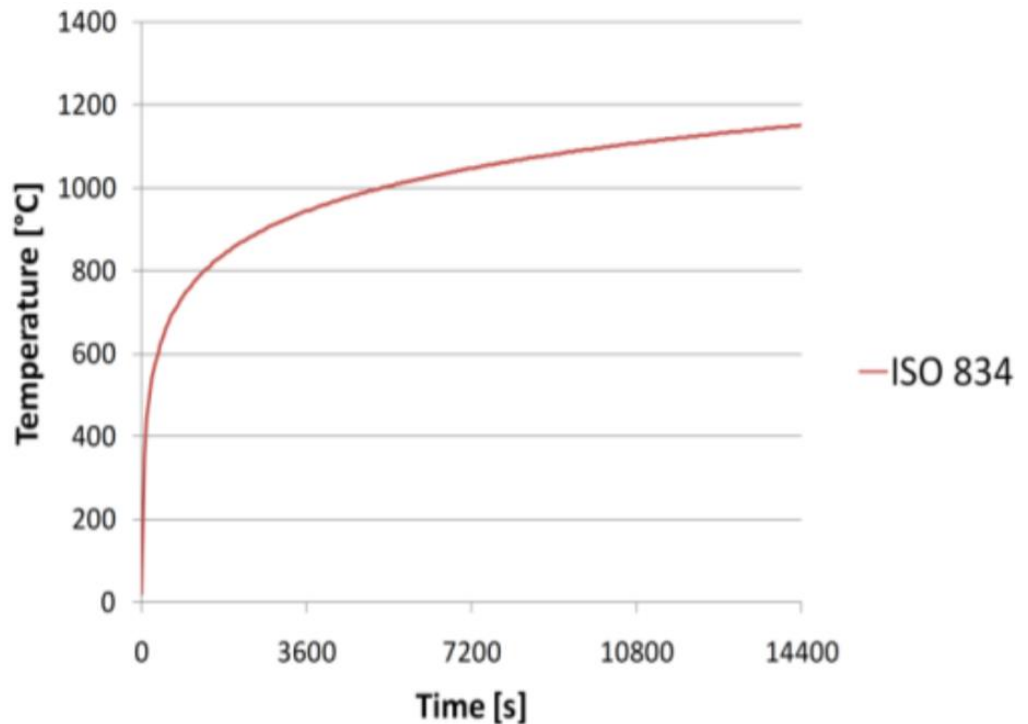
The status bar at the bottom indicates: No Messages, No Selection, Metric (mm, kg, N, s, mV, mA), Degrees, rad/s, Celsius.

Tabulkové zadání teplot

Vytvořit si v tabulce hodnoty podle rovnice.

Min	Sec	Tep (°C)
1	60	351
10	600	680
20	1200	783
30	1800	844
40	2400	887
50	3000	920
60	3600	947
70	4200	970
80	4800	990
90	5400	1008
100	6000	1024
110	6600	1038
120	7200	1051

$$T = T_0 + 345 \log_{10} (8t+1)$$



Transient Thermal – Insert Temperature

Context A : Transient Thermal - Mechanical [Ansys Mechanical Enterprise]

File Home Environment Display Selection Automation Quick Launch

Isometric Previous Rotate +Sx Rotate -Sx Pan Up Pan Down Random
 Look At Next Rotate +Sy Rotate -Sy Pan Left Pan Right Rescale
 Views Angle 10 Rotate +Sz Rotate -Sz Zoom In Zoom Out Preferences Annotation
 Display Show Mesh Thick Shells and Beams Style Cross Section Display Style
 Show Vertices Close Vertices 3, (Auto Scale) Vertex
 Edge Explode Viewports Show Display

Outline Details of "Temperature"

Name Search Outline

Project*
 Model (A4)
 Geometry
 Materials
 Coordinate Systems
 Mesh
 Transient Thermal (A5)
 Initial Temperature
 Analysis Settings
 Temperature
 Solution (A6)

Details of "Temperature"
 Scope
 Scoping Method Geometry Selection
 Geometry 1 Face
 Definition
 Type Temperature
 Magnitude Tabular Data
 Suppressed No
 Tabular Data
 Independent Variable Time

A: Transient Thermal
 Temperature
 Time: 7200, s
 25.11.2021 12:54
 Temperature: 1051, °C

ANSYS 2020 R2

0,00 600,00 (mm)
 300,00

Messages Tabular Data

Text

Steps	Time [s]	Temperature [°C]
1	0,	351,
2	1,	351,
3	600,	680,
4	1200,	783,
5	1800,	844,
6	2400,	887,
7	3000,	920,
8	3600,	947,
9	4200,	970,
10	4800,	990,
11	5400,	1008,
12	6000,	1024,
13	6600,	1038,
14	7200,	1051,

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

Solution - Temperature

ANSYS 2020 R2

A: Transient Thermal - Mechanical [Ansys Mechanical Enterprise]

File Home Result Display Selection Automation

Quick Launch

0, (Auto Scale)
Scoped Bodies
 Large Vertex Contours

Geometry Contours Edges
Display

Probe
Maximum
Minimum
Vector Display
Capped Isosurface
Views

Outline

Name Search Outline

Project*

- Model (A4)
 - Geometry
 - Materials
 - Coordinate Systems
 - Mesh
- Transient Thermal (A5)
 - Initial Temperature
 - Analysis Settings
 - Temperature
- Solution (A6)
 - Solution Information
 - Temperature - Global Ma
 - Temperature - Global Mir
 - Temperature
 - Total Heat Flux

Details of "Temperature"

Scope

Scoping Method	Geometry Selection
Geometry	All Bodies

Definition

Type	Temperature
By	Time
<input type="checkbox"/> Display Time	7200, s
Calculate Time History	Yes
Identifier	
Suppressed	No

Results

<input type="checkbox"/> Minimum	958,62 °C
<input type="checkbox"/> Maximum	1051, °C
<input type="checkbox"/> Average	1002,3 °C
Minimum Occurs On	SYS\Solid
Maximum Occurs On	SYS\Solid

Minimum Value Over Time

<input type="checkbox"/> Minimum	22,684 °C
<input type="checkbox"/> Maximum	958,62 °C

Maximum Value Over Time

<input type="checkbox"/> Minimum	390, °C
<input type="checkbox"/> Maximum	1051, °C

Information

A: Transient Thermal
Temperature
Type: Temperature
Unit: °C
Time: 7200
25.11.2021 12:57

1051 Max
1040,7
1030,5
Automatic
1009,9
999,68
989,41
979,15
968,88
958,62 Min

0,00 600,00 (mm)
300,00

Graph

Animation 20 Frames

7200

1051, 750, 500, 250, 22,684

0, 1000, 2000, 3000, 4000, 5000, 7200, [s]

Tabular Data

Time [s]	Minimum [°C]	Maximum [°C]
1	72, 22,684	390,
2	144, 25,043	429,54
3	198,45	28,131 459,45
4	252,89	32,702 489,35
5	416,23	57,838 579,06
6	906,23	173,85 732,57
7	1626,2	345,01 826,33
8	2346,2	491,6 883,15
9	3066,2	610,7 922,98
10	3786,2	705,9 954,14
11	4506,2	781,9 980,21
12	5226,2	842,94 1002,8
13	5946,2	892,4 1022,6
14	6666,2	932,79 1039,4

Ready

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

Static Structural – mech 25 mm

The screenshot displays the ANSYS 2020 R2 software interface for a Static Structural analysis. The main window shows a 3D model of a mechanical part with a mesh overlay. The 'Details of "Mesh"' panel is open, showing the following settings:

Category	Setting
Display	Display Style: Use Geometry Setting
Defaults	Physics Preference: Mechanical
	Element Order: Program Controlled
	Element Size: 25, mm
Sizing	
Quality	
Inflation	
Advanced	
Statistics	

The main view shows a 3D model of a mechanical part with a mesh overlay. A scale bar indicates 0,00 to 900,00 (mm) with a midpoint at 450,00. The status bar at the bottom shows 'Ready' and 'Messages pane No Selection Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius'.

Static Structural – okrajové podmínky

The screenshot displays the ANSYS 2020 R2 Static Structural interface. The main window shows a 3D model of a beam with a red top surface and a grey bottom surface. The beam is supported by a fixed support (D) at the left end. A force (B) is applied to the top surface, and a displacement (C) is applied to the right end. The beam is oriented along the X-axis, with a scale bar indicating a length of 600.00 mm. The Y and Z axes are shown in the bottom right corner.

Details of "Static Structural (B5)"

Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL

Options	
<input type="checkbox"/> Environment Temperature	22, °C
<input type="checkbox"/> Generate Input Only	No

B: Static Structural
Static Structural
Time: 1, s
25.11.2021 16:31

- A** Standard Earth Gravity: 9806,6 mm/s²
- B** Force: 50000 N
- C** Displacement
- D** Fixed Support

Fixed support
Displacement Z=0
Standard Gravity
Force 50 kN (+Z)

Solution – Total Deformation

ANSYS 2020 R2

B: Static Structural
 Total Deformation
 Type: Total Deformation
 Unit: mm
 Time: 1
 25.11.2021 14:16

0,31885 Max

0,28342
0,24799
0,21257
0,17714
0,14171
0,10628
0,07085
0

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

Graph

Animation [mm] [s]

Tabular Data

Time [s]	Minimum [mm]	Maximum [mm]
1 1.	0.	0,31885

Messages Graph

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

Solution do Setup

The screenshot displays the ANSYS Workbench interface for a project named "fire2 - Workbench". The main area shows a "Project Schematic" with two analyses, A and B, connected by lines indicating data flow.

Analysis A: Transient Thermal

- 1. Transient Thermal
- 2. Engineering Data ✓
- 3. Geometry ✓
- 4. Model ✓
- 5. Setup ✓
- 6. Solution ✓
- 7. Results ✓

Analysis B: Static Structural

- 1. Static Structural
- 2. Engineering Data ✓
- 3. Geometry ✓
- 4. Model ✓
- 5. Setup ⚡
- 6. Solution ⚡
- 7. Results ⚡

The "Properties of Project Schematic" panel on the right shows a table with columns A and B:

	A	B
1	Property	Value
2	Notes	
3	Notes	
4	Solution Process	
5	Update Option	R...

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

Import load – Imported body

Context | B: Static Structural - Mechanical [Ansys Mechanical Enterprise]

File | Home | Environment | Display | Selection | Automation

Quick Launch

My Computer | Distributed | Cores 2 | Solve

Named Selection | Commands | Images | Section Plane | Annotation | Tools | Layout

Analysis | Coordinate System | Comment | Section Plane | Annotation | Tools | Layout

Insert | Chart

Outline | Details of "Imported Body Temperature" | Select | Mode

Project*

- Model (B4)
 - Geometry
 - Materials
 - Coordinate Systems
 - Mesh
 - Static Structural (B5)
 - Analysis Settings
 - Standard Earth Gravi
 - Fixed Support
 - Displacement
 - Force
 - Imported Load (A6)
 - Imported Body
 - Solution (B6)
 - Solution Inform
 - Total Deformati

Details of "Imported Body Temperature"

Scope

Scoping Method	Geometry Selection
Geometry	1 Body

Definition

Type	Imported Body Temperature
Tabular Loading	Program Controlled
Suppressed	No
Source Bodies	Automatic
Tolerance	1, %
Source Time	Worksheet

Settings

Mapping Control	Program Controlled
Mapping	Profile Preserving
Weighting	Shape Function
Transfer Type	Volumetric

Rigid Transformation

Mesh Alignment	Use Origin and Euler Angles
Origin X	0, mm
Origin Y	0, mm
Origin Z	0, mm
Theta XY	0, degree
Theta YZ	0, degree
Theta ZX	0, degree

Graphics Controls

Display Source Points	Off
Display Source Point Ids	Off

Legend Controls

Legend Range	Program Controlled
Minimum Source	958,62 °C
Maximum Source	1051, °C

Named Selection Creation

Unmapped Nodes	Off
Mapped Nodes	Off
Outside Nodes	Off

B: Static Structural

Imported Body Temperature
Time: 1, s
Unit: °C
25.11.2021 14:17

1051 Max

1040,7
1030,5
1020,2
1009,9
999,68
989,41
979,15
Min
958,62 Min

Imported Body Temperature

Max

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

Data View

Imported Body Temperature

Steps	Time [s]	Temperature
1	1	Row 1

Source Time (s) Analysis Time (s)

Source Time (s)	Analysis Time (s)
1	End Time
1	1

Messages | Graph | Data View

(8%) Creating solver input file... | No Messages | No Selection | Metric (mm, kg, N, s, mV, mA) | Degrees | rad/s | Celsius

Deformation

Context | B: Static Structural - Mechanical [Ansys Mechanical Enterprise]

File Home Result Display Selection Automation

Quick Launch

21 (5x Auto) | Scoped Bodies | Large Vertex Contours | Geometry | Contours | Edges | Probe | Maximum | Minimum | Vector Display | Capped Isosurface | Views

Outline | Details of "Total Deformation" | Select Mode

Project*

- Model (B4)
 - Geometry
 - Materials
 - Coordinate Systems
 - Mesh
- Static Structural (B5)
 - Analysis Settings
 - Standard Earth Gravity
 - Fixed Support
 - Displacement
 - Force
 - Imported Load (A6)
 - Imported Body Temper
- Solution (B6)
 - Solution Information
 - Total Deformation

Details of "Total Deformation"

Scope

Scoping Method	Geometry Selection
Geometry	All Bodies

Definition

Type	Total Deformation
By	Time
<input type="checkbox"/> Display Time	Last
Calculate Time History	Yes
Identifier	
Suppressed	No

Results

<input type="checkbox"/> Minimum	0, mm
<input type="checkbox"/> Maximum	36,493 mm
<input type="checkbox"/> Average	17,953 mm
<input type="checkbox"/> Minimum Occurs On	SYS\Solid
<input type="checkbox"/> Maximum Occurs On	SYS\Solid

Information

B: Static Structural
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1
25.11.2021 14:19

ANSYS 2020 R2

36,493 Max
32,438
28,383
24,328
20,274
16,219
12,164
8,1095
4,0547
0 Min

0,00 250,00 500,00 (mm)

Graph | Animation | 20 Frames

Tabular Data

Time [s]	Minimum [mm]	Maximum [mm]	Average [mm]
1	0	36,493	17,953

Messages | Graph

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