

Nedjelja 12

Modeliranje toplotnog opterećenja mašinskog elementa konačnim elementima

Postavka zadatka

Elementi za odvod toplote se uobičajeno koriste kako bi se omogućila disipacija toplote koju generišu elektronski uređaji. Element za odvod toplote je izrađen od aluminijuma (toplotna konduktivnost $k=170 \text{ W}/(\text{m}\cdot\text{K})$, specifična toplota $c=870 \text{ J}/(\text{kg}\cdot\text{K})$, $E=70 \text{ GPa}$, $\nu=0.3$, $\rho=2800 \text{ kg}/\text{m}^3$, koeficijent toplotnog širenja $\alpha=22\cdot 10^{-6} \text{ 1}/^\circ\text{C}$). Ventilacijom vazduha po svim površinama, izuzev donje površine, elementa za odvod toplote se pospješuje hlađenje uređaja čiji je ulazni toplotni fluks q .

Postavka zadatka

Vazduh u radnom prostoru elementa za odvod toplote je temperature od $T=28\text{ }^{\circ}\text{C}$ sa koeficijentom prenosa toplote $h=30\text{ W}/(\text{m}^2\cdot\text{ }^{\circ}\text{C})$, .
Odrediti:

- Raspodjelu temperature pri stacionarnom stanju ukoliko je početna temperature elementa za odvod toplote $T=28\text{ }^{\circ}\text{C}$, konstatni ulazni toplotni fluks po donjoj površini $q=1000\text{ W}/\text{m}^2$

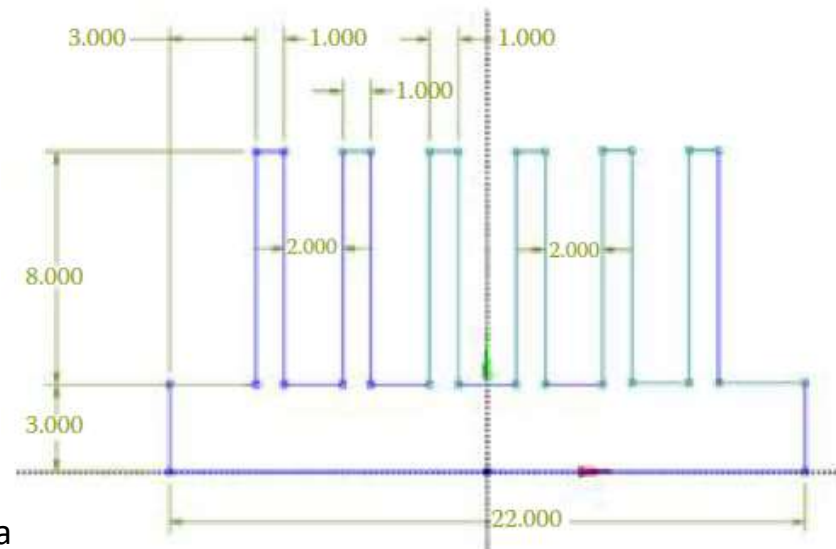
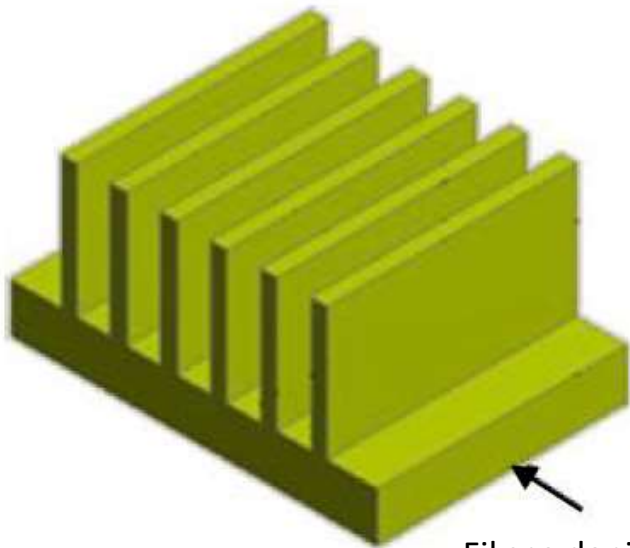
Postavka zadatka

- Raspodjelu temperature pri nestacionarnom stanju ukoliko je ulazni toplotni fluks kvadratna funkcija sa periodom $\tau=90$ s i minimalnom, odnosno, maksimalnom veličinom $q=(0\div 1000)$ W/m². Početno stanje raspodjele temperature odgovara raspodjeli dobijenoj u prethodnom koraku. Promjenu raspodjele temperature pratiti $t=180$ s.
- Raspodjelu toplotnih napona u stacionarnom stanju.

Postavka zadatka

Granični uslovi: Nepokretan oslonac po donjoj površini

Opterećenje: Toplotni fluks po donjoj površini



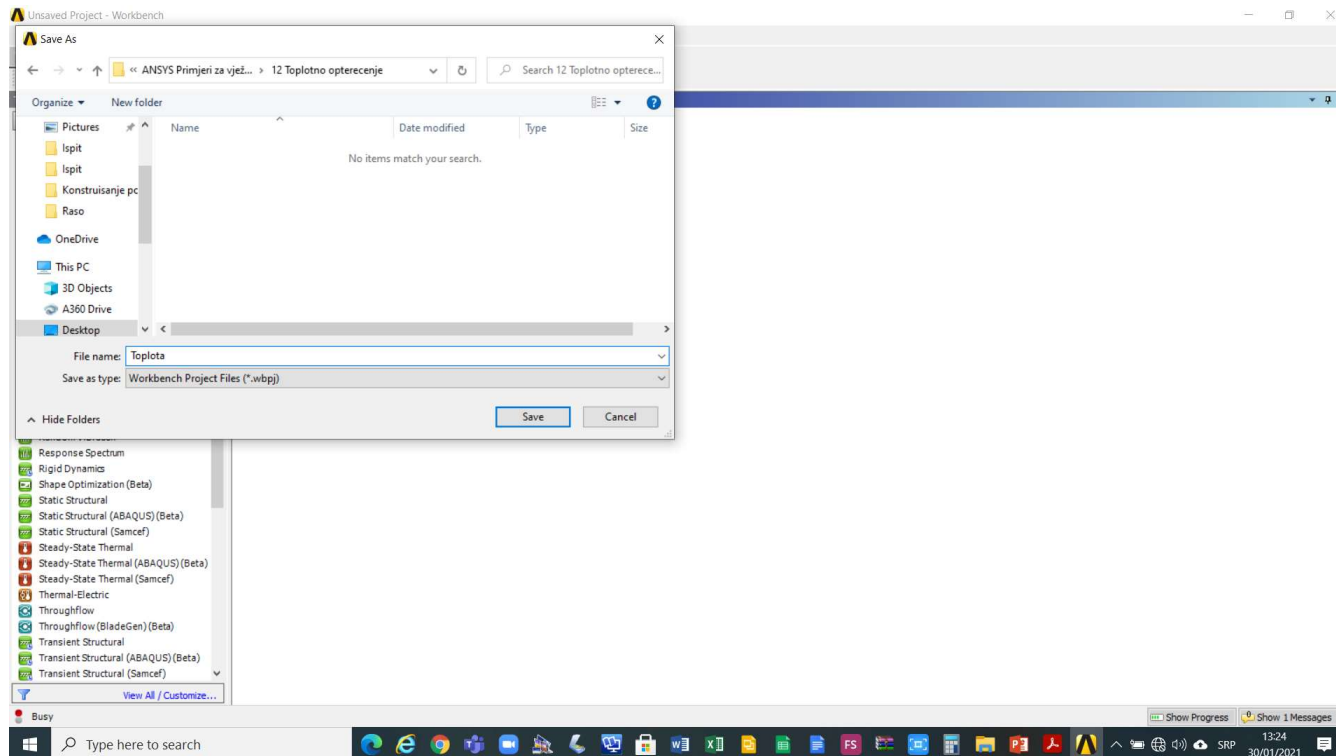
Sve dimenzije su date u mm

Postavka zadatka



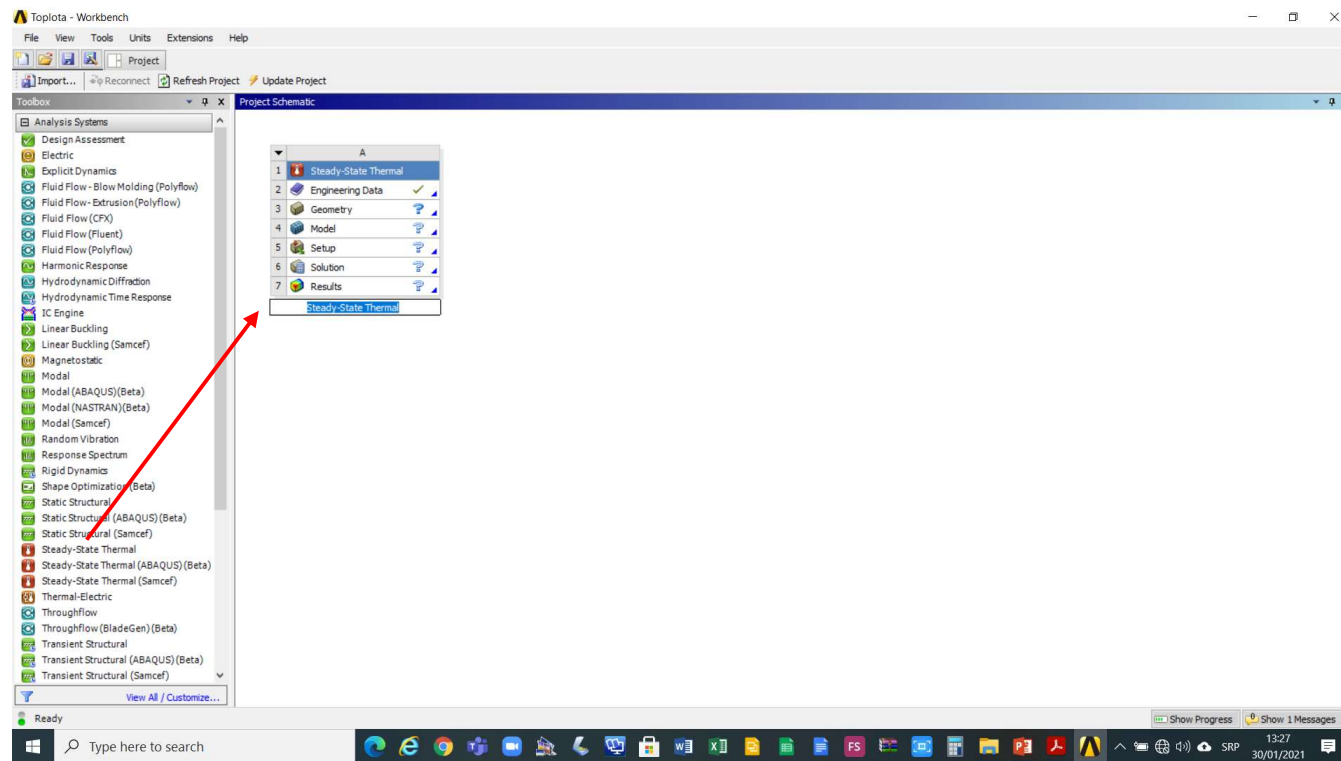
Toplotno opterećenje

Aktivirati program ANSYS i sačuvati prazan projekat pod nazivom Toplota



Toplotno opterećenje

Kreirati stacionarnu termalnu analizu (*Steady State Thermal*) na shemi projekta (*Project Schematic*)



Toplotno opterećenje

Aktivirati modul Engineering Data (*Engineering Data->Edit*) i dodati novi materijal *Click here to add new material*

The screenshot displays the ANSYS Workbench Engineering Data interface. On the left, the 'Engineering Data' component is selected, and the 'Edit...' menu is open, showing options like 'Duplicate', 'Transfer Data From New', 'Update', 'Refresh', 'Rename', 'Properties', 'Quick Help', and 'Add Note'. The main window shows the 'Properties of Outline Row 3: Structural Steel' table, which lists material properties such as Density (7850 kg m⁻³), Young's Modulus (2E+11 Pa), and Poisson's Ratio (0.3). Below this is the 'Outline of Schematic A2: Engineering Data' table, which includes a 'Material' section with 'Structural Steel' and a 'Click here to add a new material' button. To the right, there are two smaller windows: 'Table of Properties Row 2: Density' and 'Chart of Properties Row 2: Density'. The chart shows a plot of Density (kg m⁻³) versus Temperature (C), with a single data point at 7850 kg m⁻³ at 0 C.

Property	Value	Unit
Density	7850	kg m ⁻³
Isotropic Secant Coefficient of Thermal Expansion		
Isotropic Elasticity		
Derive from	Young's Modulus a...	
Young's Modulus	2E+11	Pa
Poisson's Ratio	0.3	
Bulk Modulus	1.6667E+11	Pa
Shear Modulus	7.6923E+10	Pa
Alternating Stress Mean Stress		
Strain-Life Parameters		
Tensile Yield Strength	2.5E+08	Pa
Compressive Yield Strength	2.5E+08	Pa
Tensile Ultimate Strength	4.6E+08	Pa
Compressive Ultimate Strength	0	Pa

Table of Properties Row 2: Density	Value
Temperature (C)	Density (kg m ⁻³)
1	7850

Outline of Schematic A2: Engineering Data	Material	Description
1	Contents of Engineering Data	Source
2	Material	
3	Structural Steel	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material	

Chart of Properties Row 2: Density	Temperature [C]	Density (kg m ⁻³)
1	0	7850

Toplotno opterećenje

Promjeniti jedinice i unijeti karakteristiku novog izotropnog materijala (*Isotropic Thermal Conductivity*)

The screenshot displays the Toplotna - Workbench software interface. The main window is titled "Outline of Schematic: AZ: Engineering Data". The "Toolbox" on the left has "Isotropic Thermal Conductivity" highlighted with a red circle. The "Outline of Schematic" table shows a hierarchy: "Contents of Engineering Data" (row 1), "Material" (row 2), "Structural Steel" (row 3), and "Aluminum" (row 4). The "Properties of Outline Row 4: Aluminum" table shows "Isotropic Thermal Conductivity" with a value of 170 and units of $W m^{-1} K^{-1}$. The "Table of Properties Row 2: Isotropic Thermal Conductivity" table shows "Temperature (C)" and "Thermal Conductivity ($W m^{-1} K^{-1}$)" with a value of 170. The "Chart of Properties Row 2: Isotropic Thermal Conductivity" shows a plot of Thermal Conductivity ($W m^{-1} K^{-1}$) vs Temperature (C) with a single data point at 170.

Outline of Schematic: AZ: Engineering Data			
1	Contents of Engineering Data	Source	Description
2	Material		
3	Structural Steel		Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
4	Aluminum		
*	Click here to add a new material		

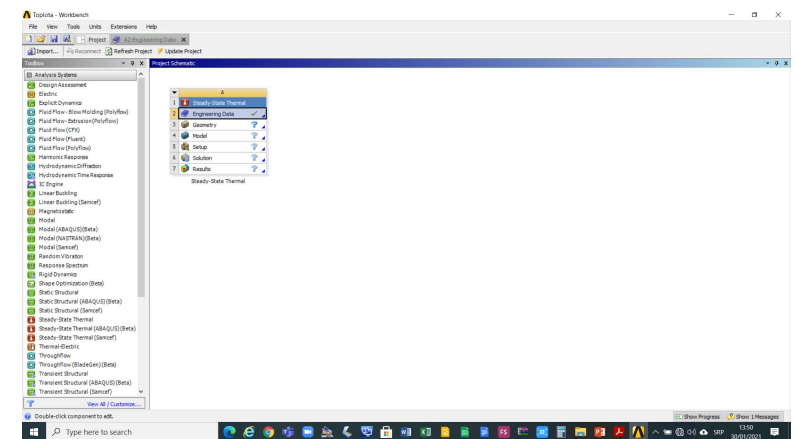
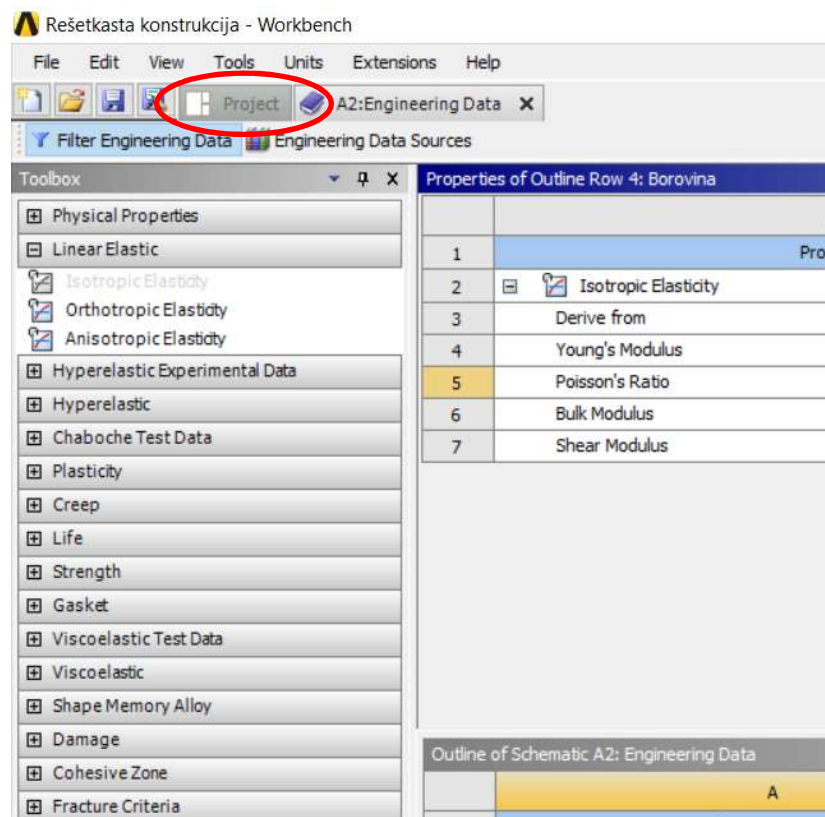
Properties of Outline Row 4: Aluminum					
1	Property	Value	Unit	D	E
2	Isotropic Thermal Conductivity	170	$W m^{-1} K^{-1}$		

Table of Properties Row 2: Isotropic Thermal Conductivity		
1	Temperature (C)	Thermal Conductivity ($W m^{-1} K^{-1}$)
2		170
*		

Chart of Properties Row 2: Isotropic Thermal Conductivity	
Thermal Conductivity ($W m^{-1} K^{-1}$)	Temperature (C)
170	0

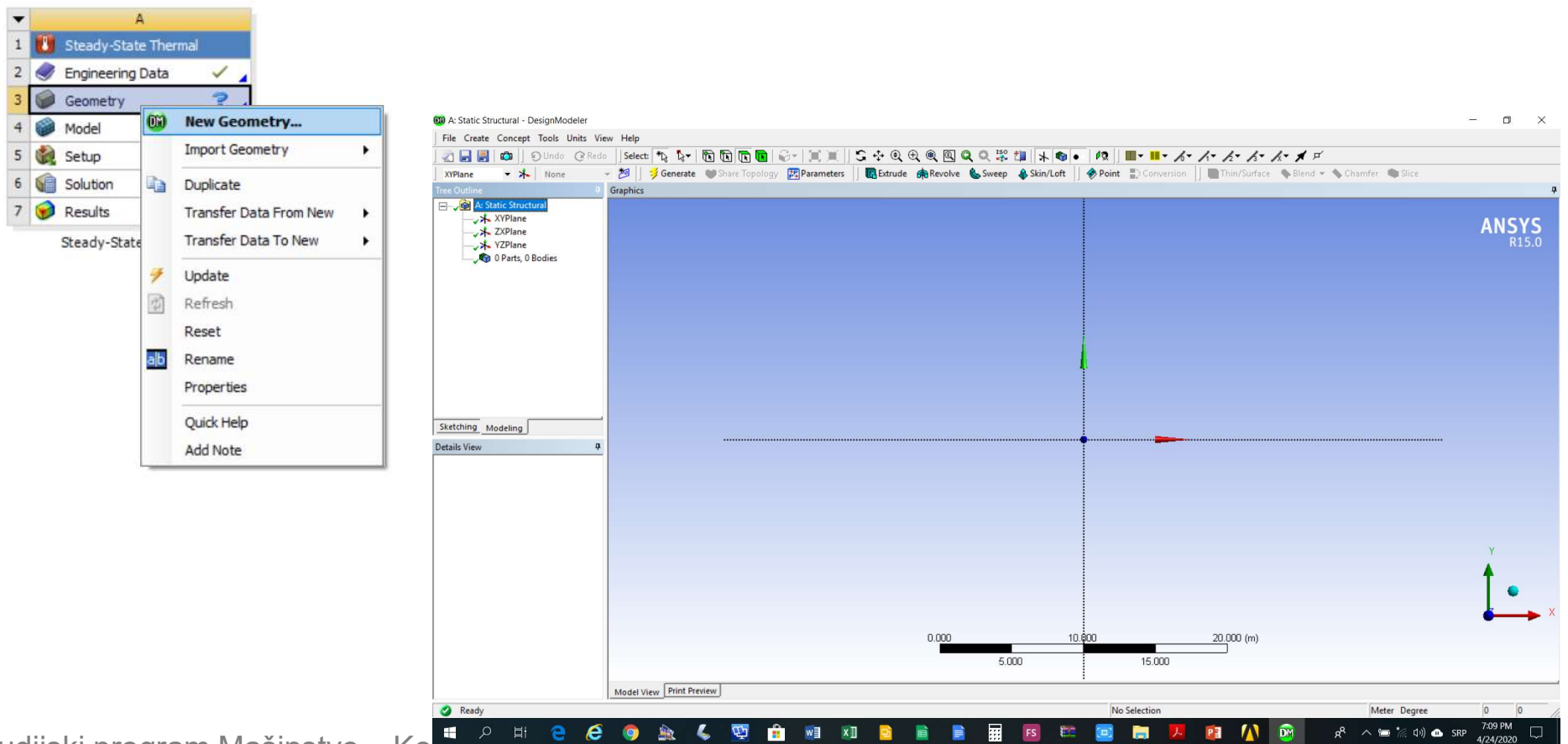
Toplotno opterećenje

Izabrati opciju *Project* za povratak na shemu projekta



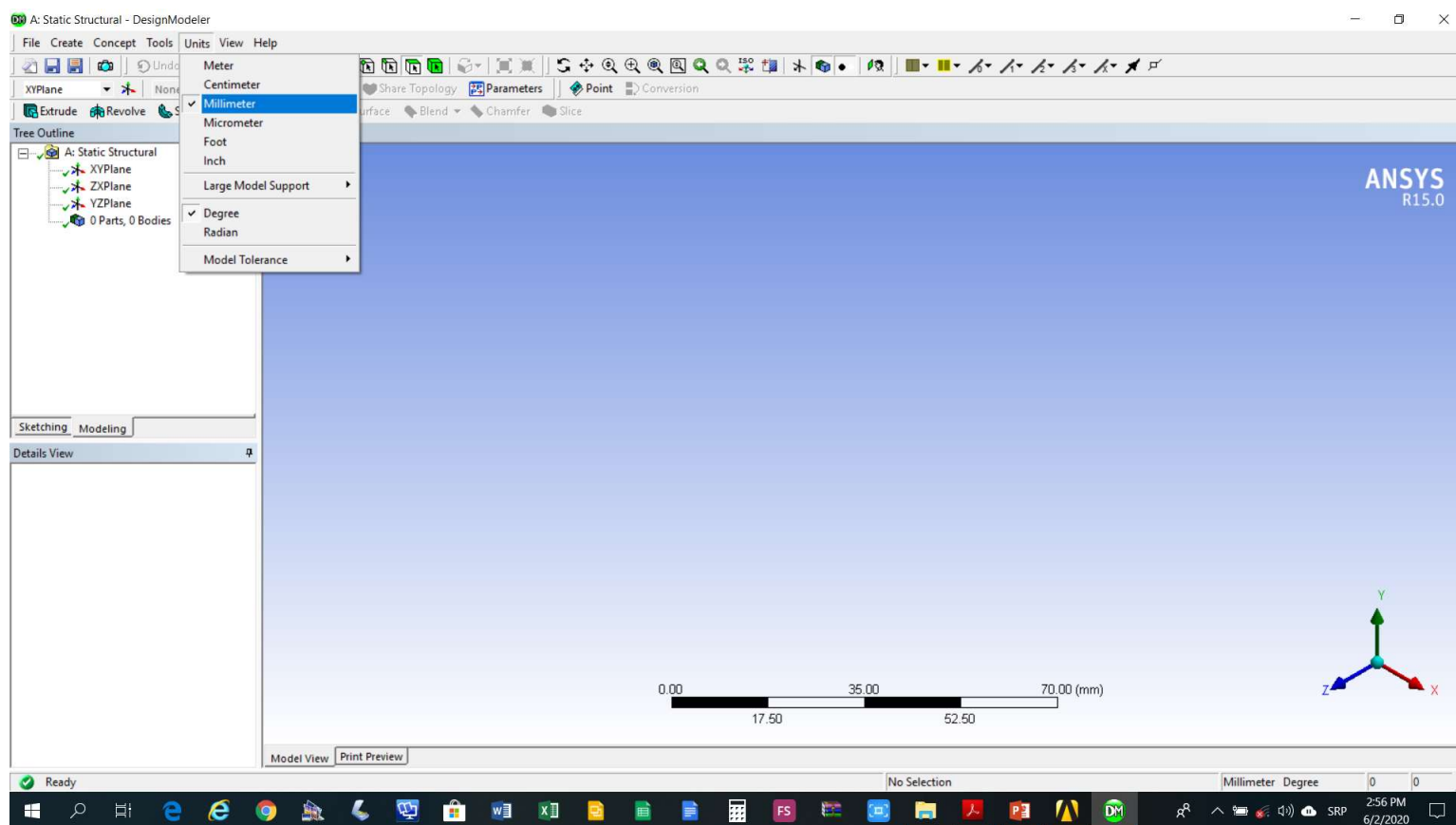
Toplotno opterećenje

Aktivirati modul Design Modeler (*Geometry->New Geometry*)



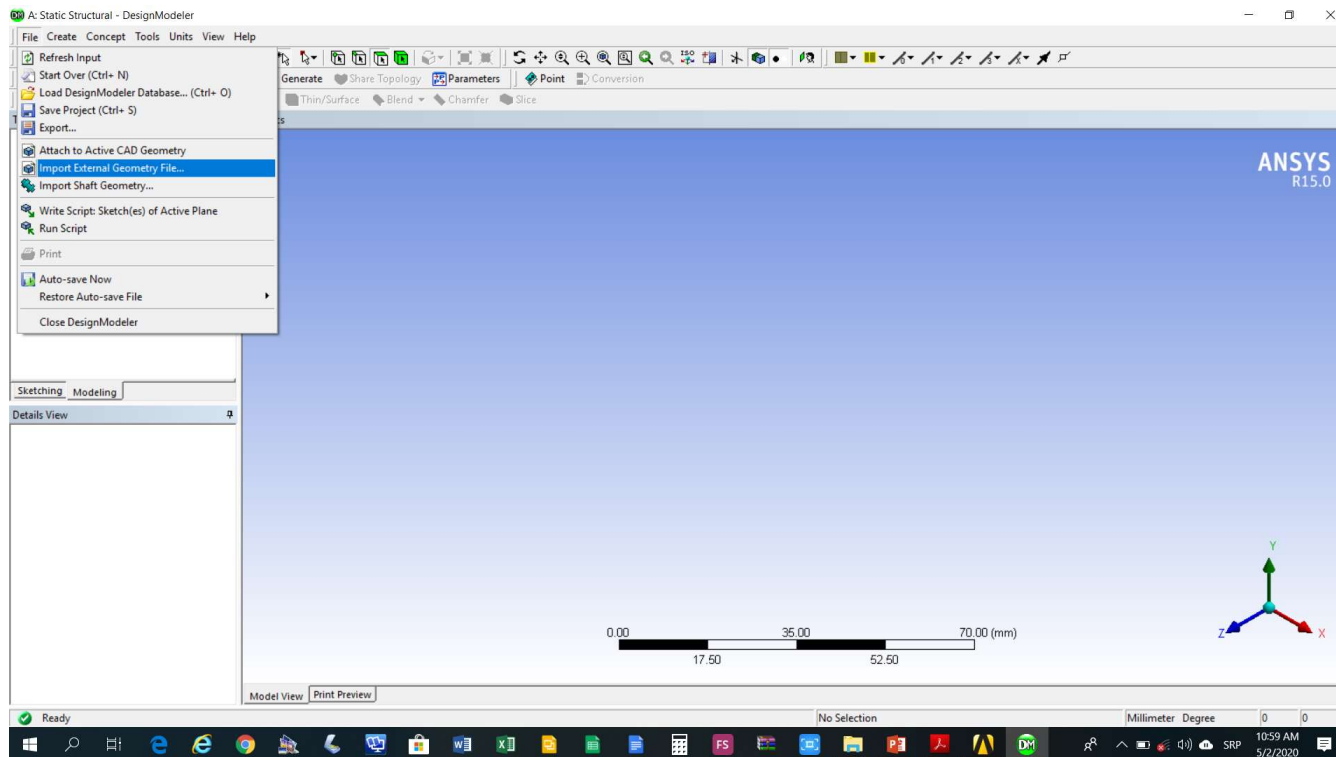
Toplotno opterećenje

Podesiti dužinske jedinice (Units->*Milimeter*)



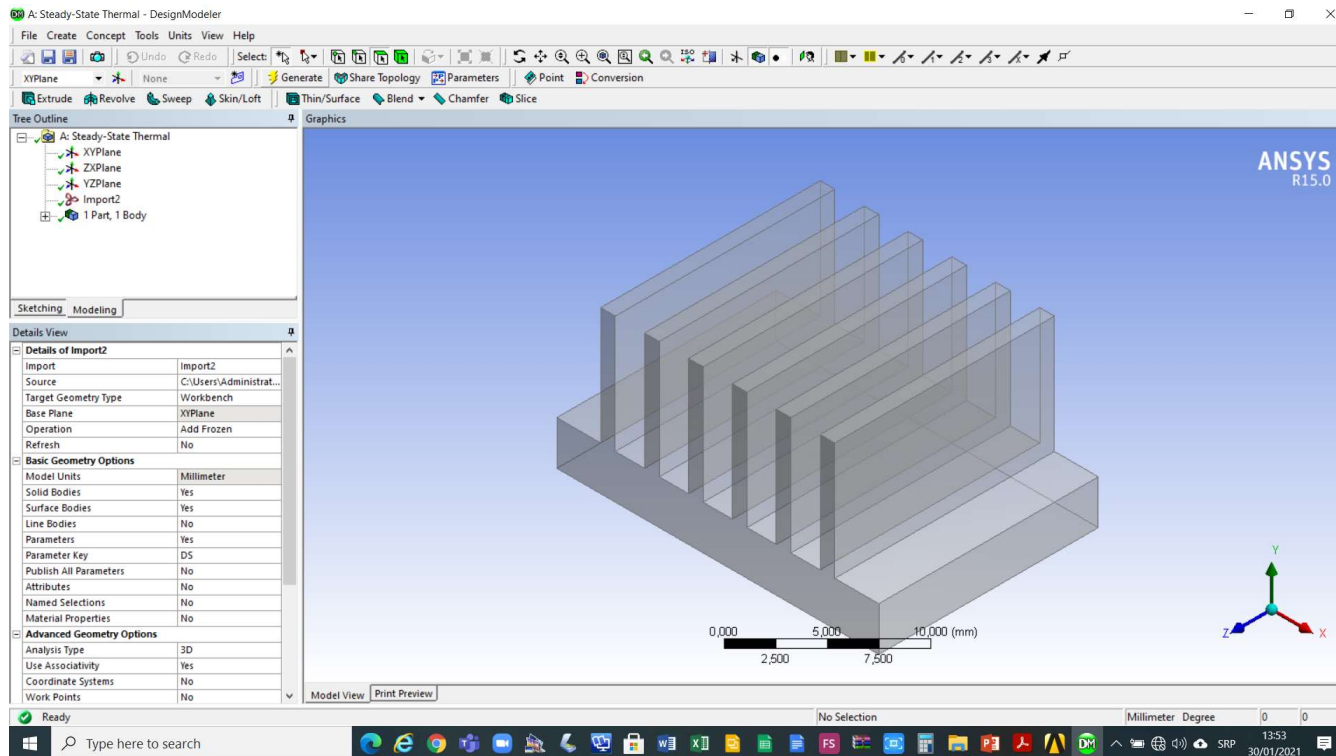
Toplotno opterećenje

Učitavanje eksterno generisane geometrije (File->*Import External Geometry File*) *.sat format



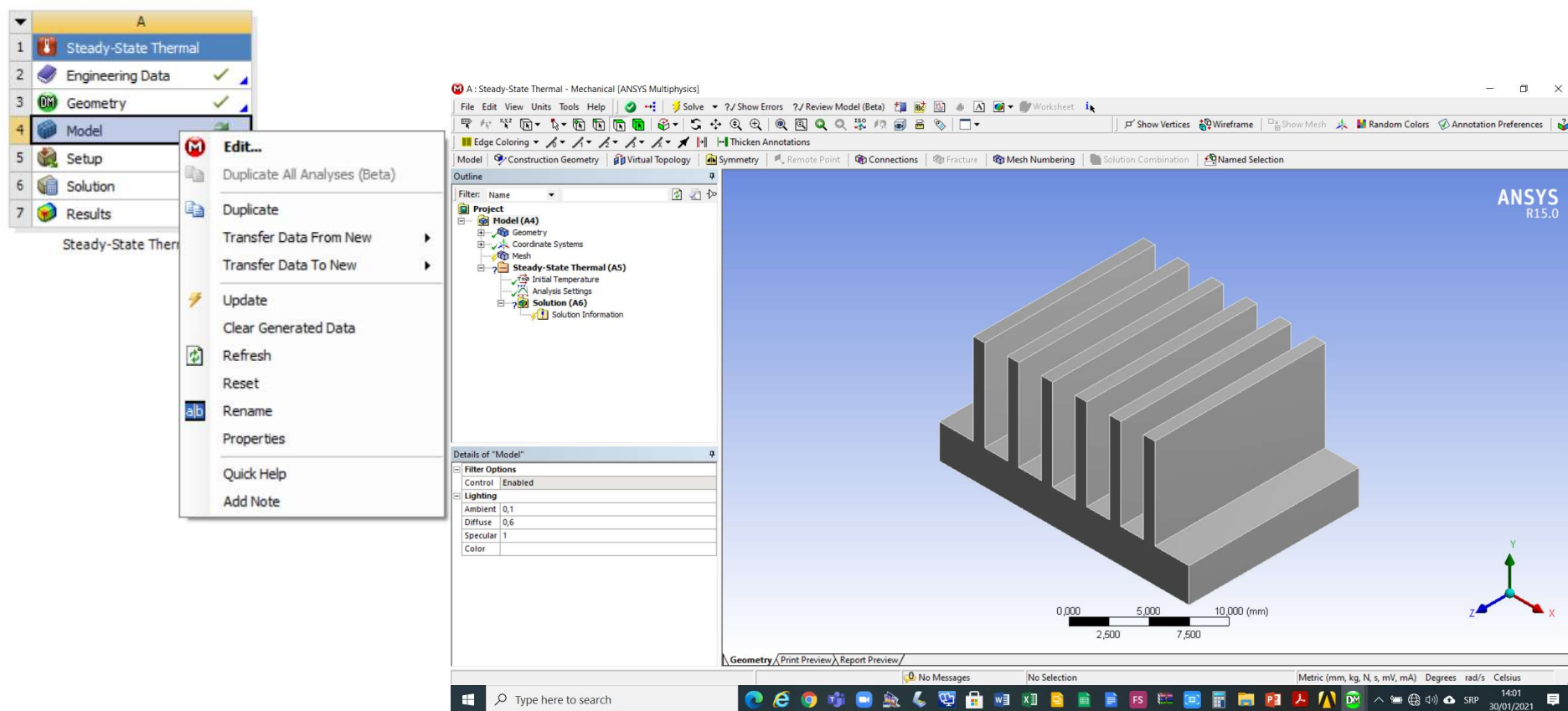
Toplotno opterećenje

Učitavanje eksterno generisane geometrije
okončati komandom *Generate*



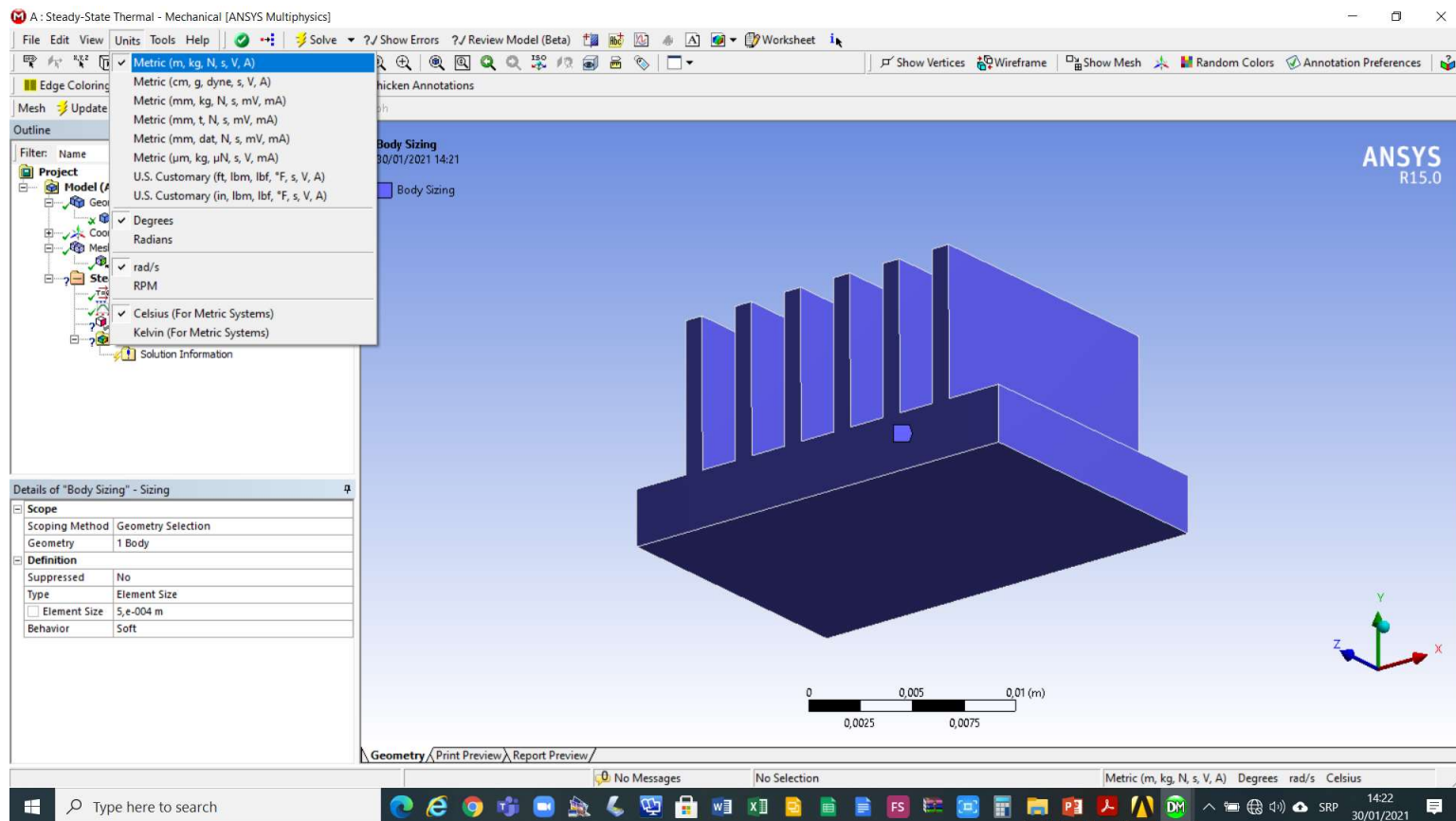
Toplotno opterećenje

Aktivirati modul Steady-State Thermal (*Model->Edit*)



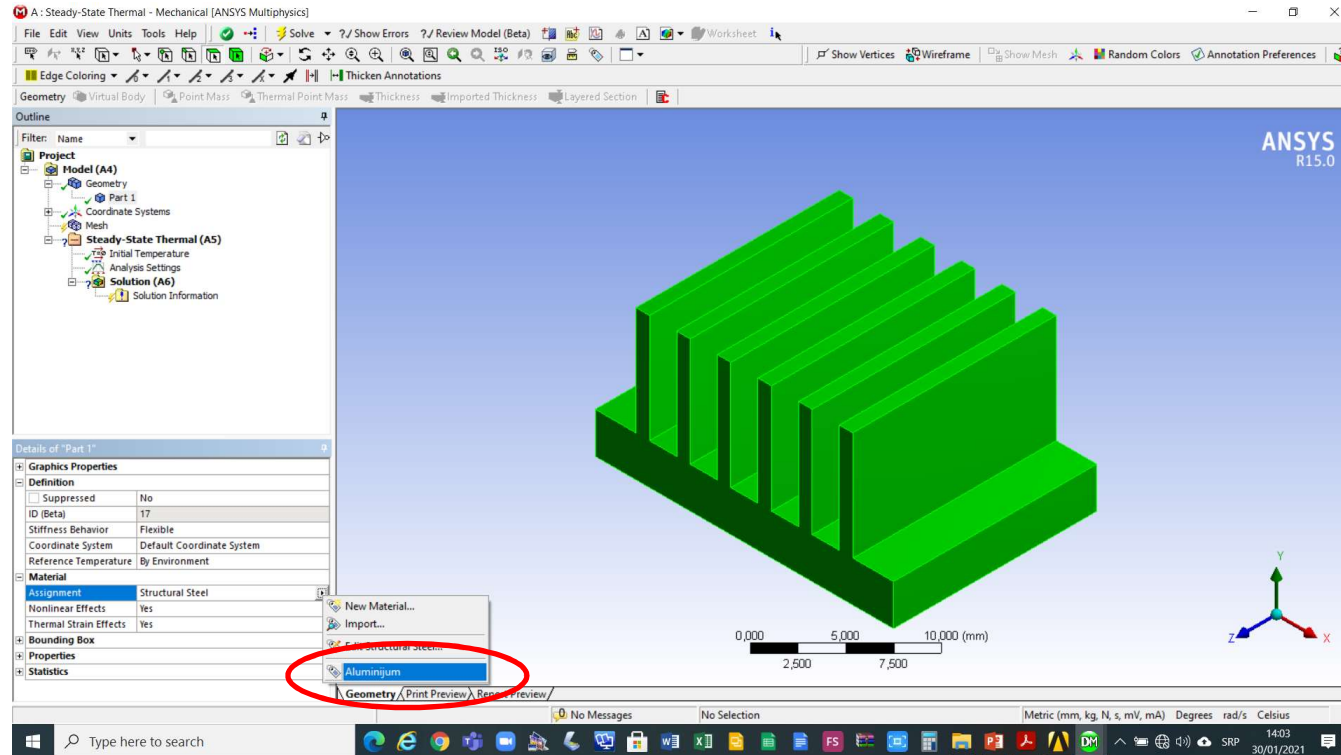
Toplotno opterećenje

Podesiti dužinske jedinice (*Units->Meter*)



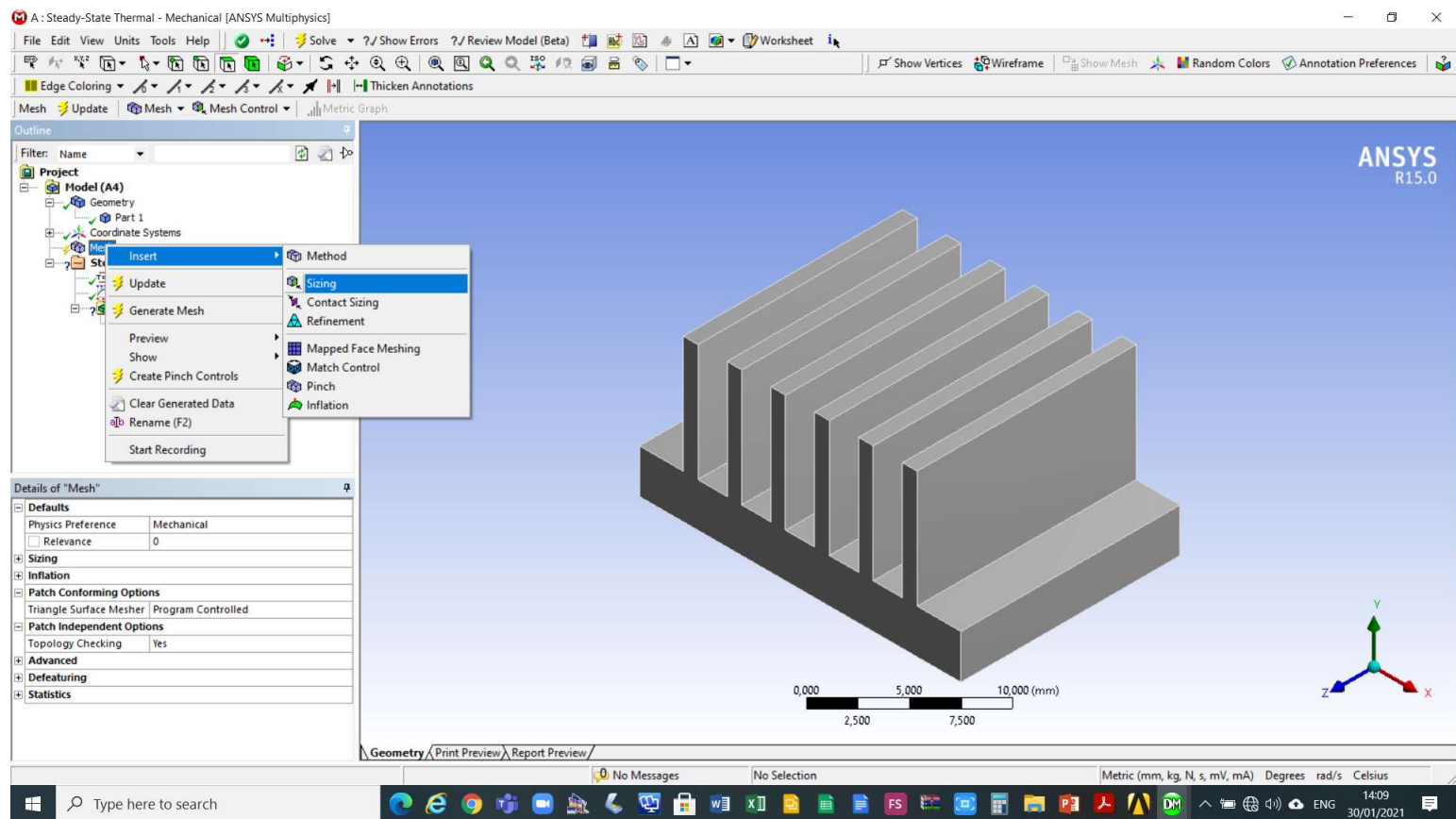
Toplotno opterećenje

Izabrati materijal *Details of Part 1->Material->Assignment* podesiti na Aluminijum



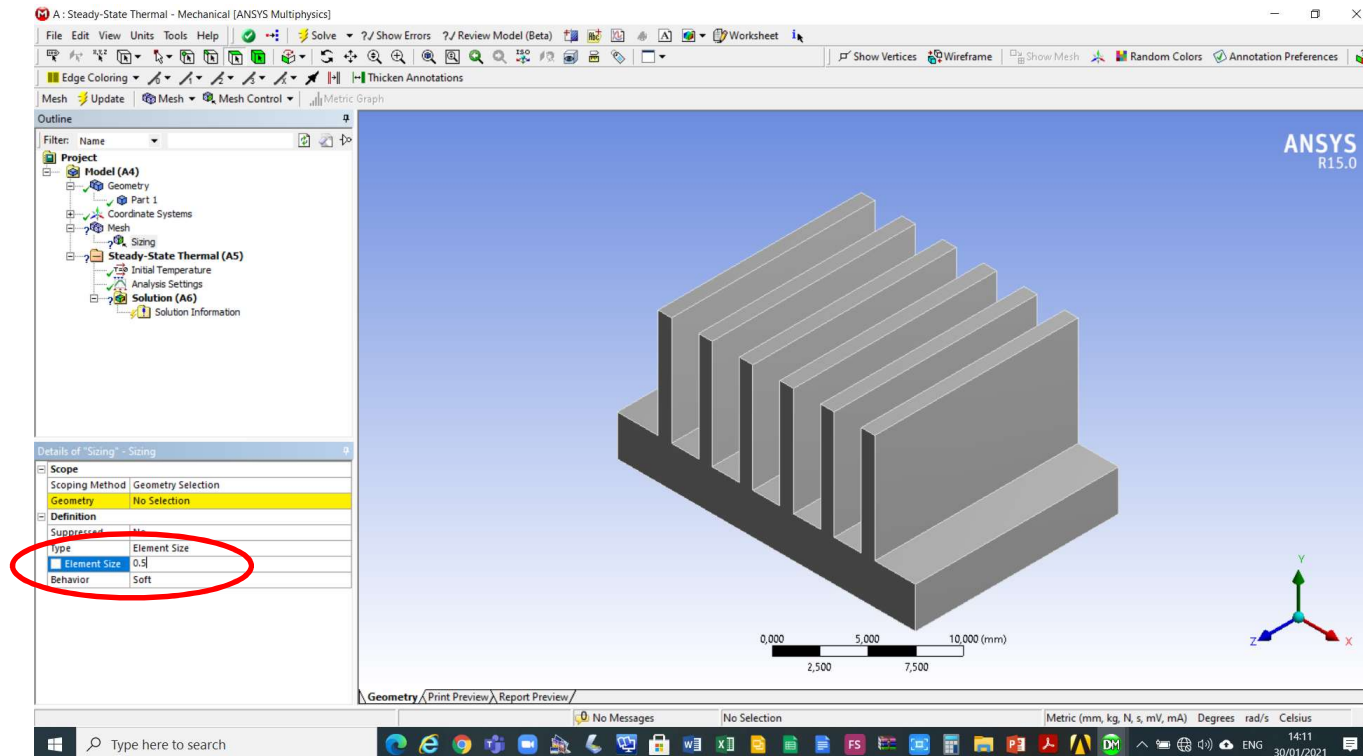
Toplotno opterećenje

Dodati stavku *Mesh->Insert->Sizing* u stablo projekta



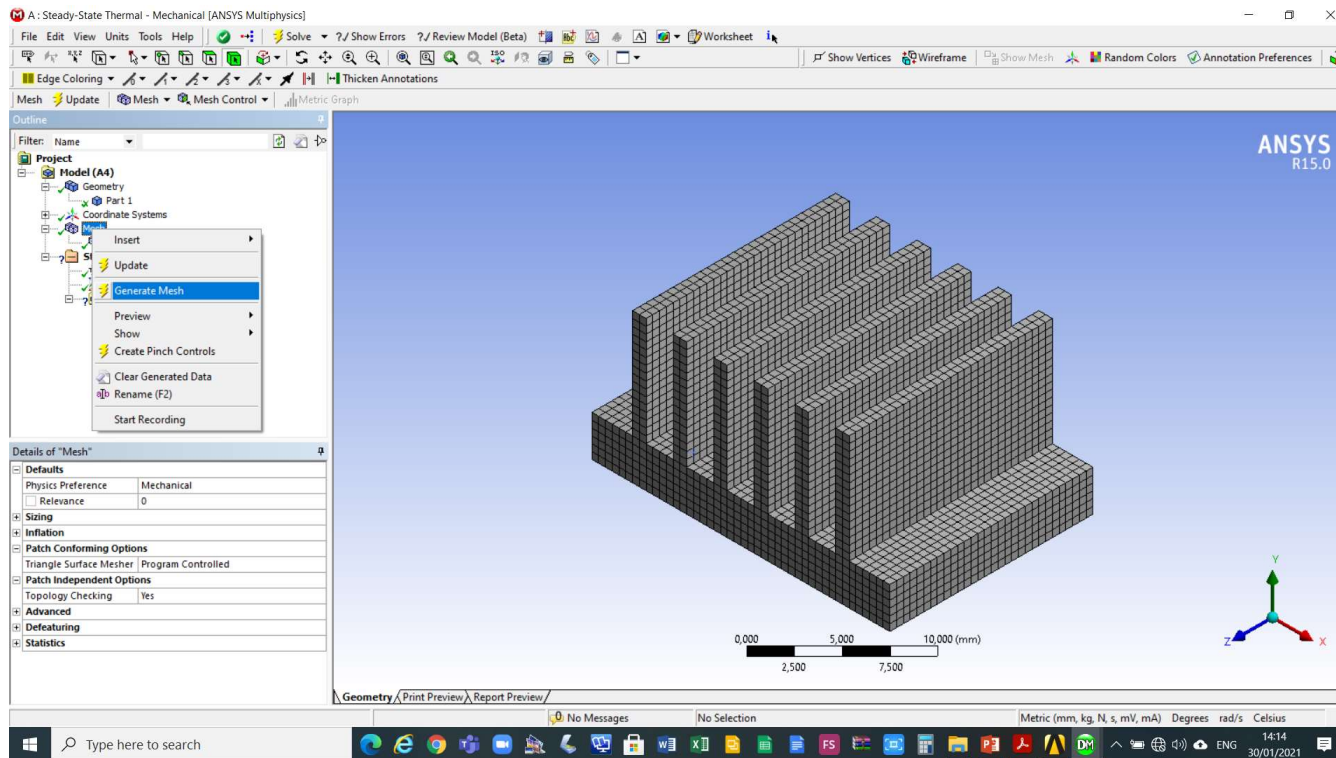
Toplotno opterećenje

Zadati veličinu konačnih elemenata *Details of Mesh->Sizing->Element Size* na $5e-4$ m



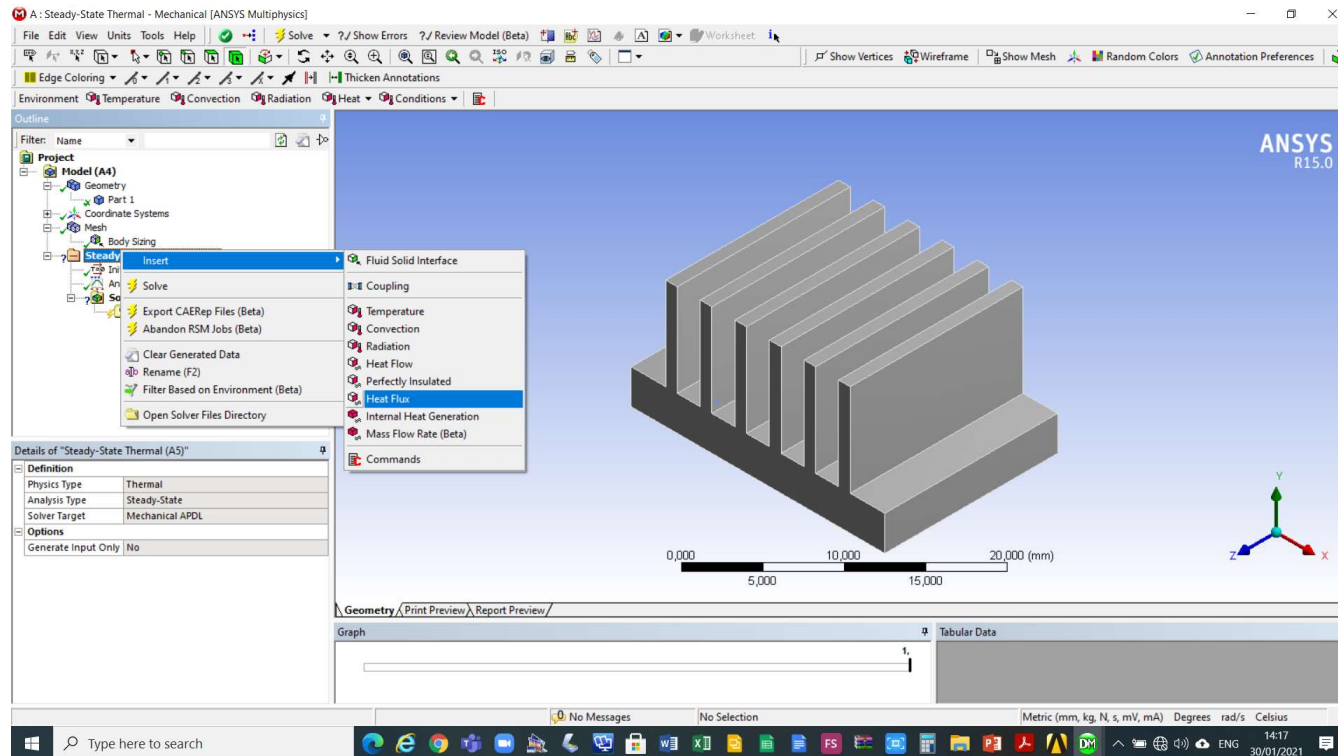
Toplotno opterećenje

Generisati mrežu konačnih elemenata *Mesh*-
>*Generate Mesh*



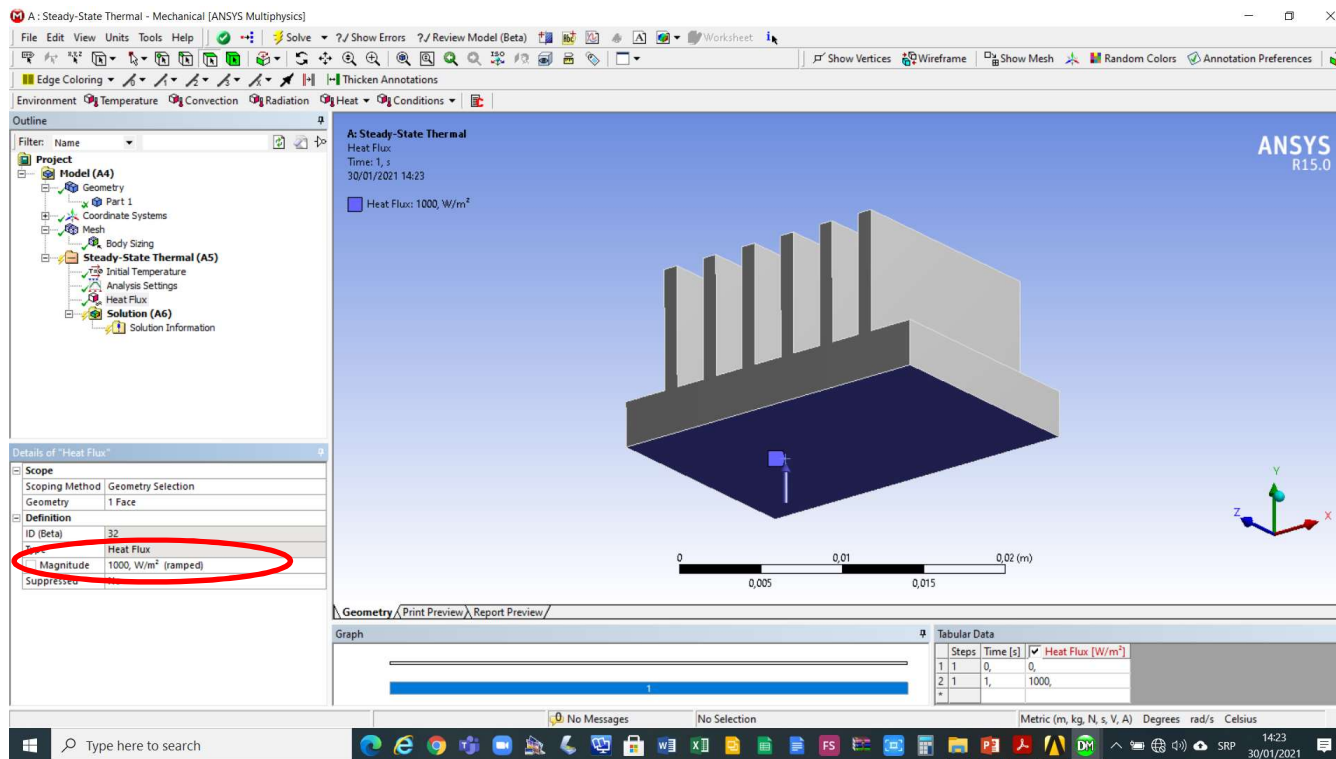
Toplotno opterećenje

Dodati toplotni fluks u model *Steady-State Thermal*->*Insert*->*Heat Flux*



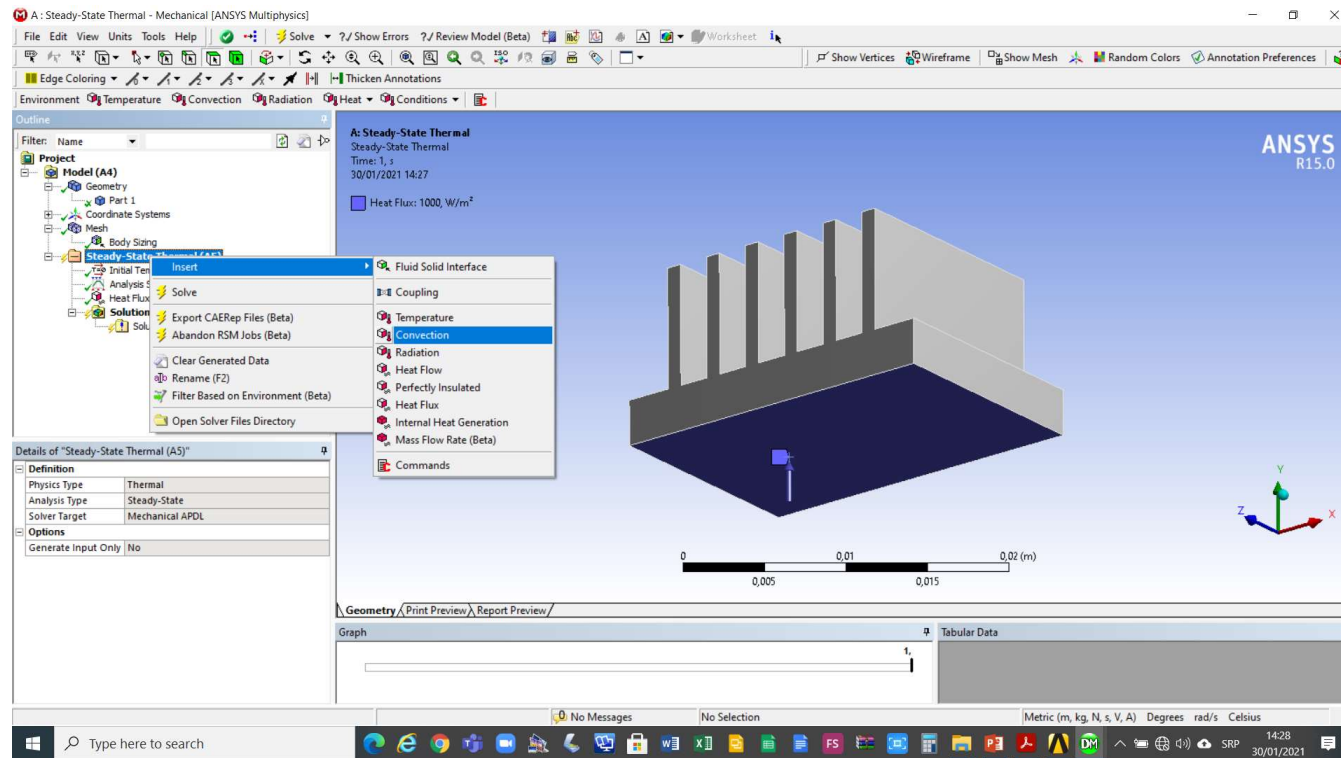
Toplotno opterećenje

Podešavanje veličine toplotnog fluksa *Details of Heat Flux*->*Definition*->*Magnitude* koji djeluje po donjoj površini na 1000 W/m^2



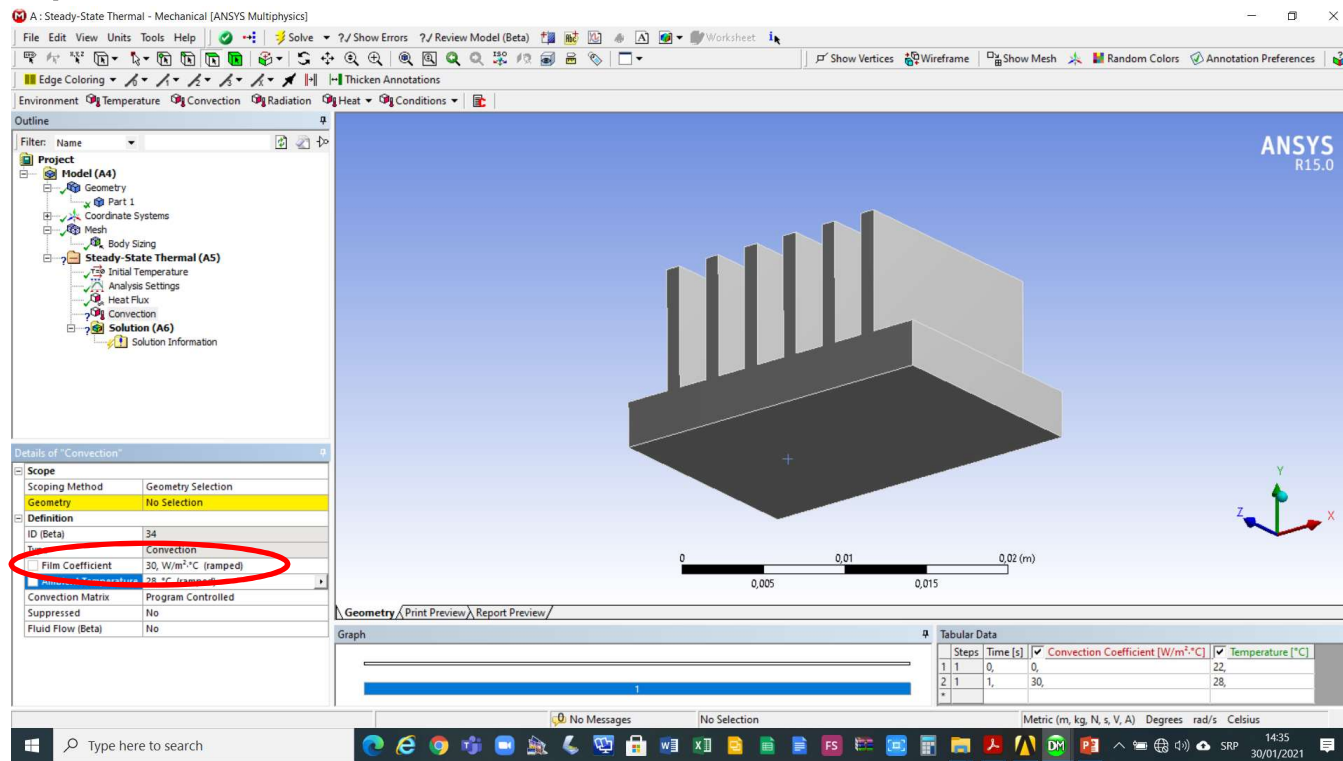
Toplotno opterećenje

Dodati konvektivni prenos toplote u model *Steady-State Thermal*->*Insert*->*Convection* koji djeluje na sve (29 površina), osim donje površine



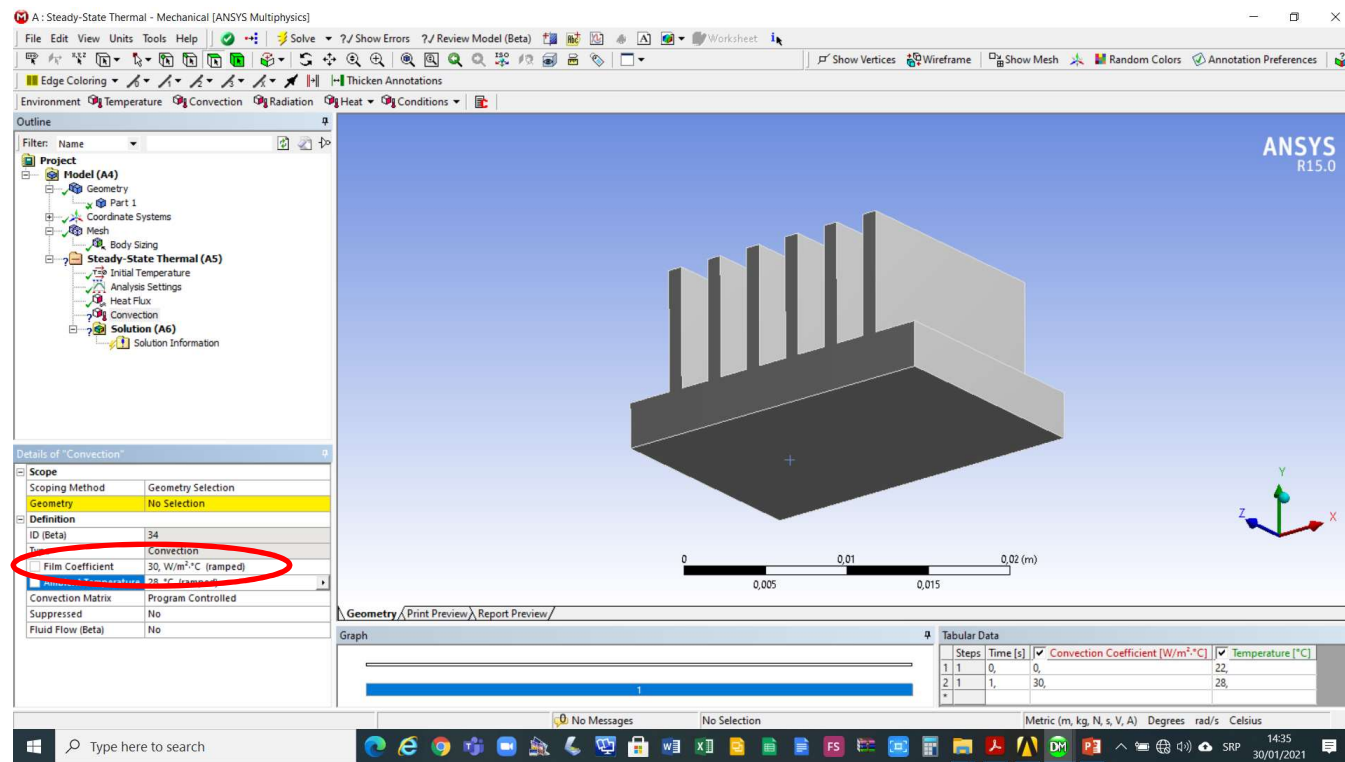
Toplotno opterećenje

Podešavanje koeficijenta prenosa toplote *Details of Convection*->*Definition*->*Film Coefficient* na $30 \text{ W}/(\text{m}^2 \cdot ^\circ\text{C})$



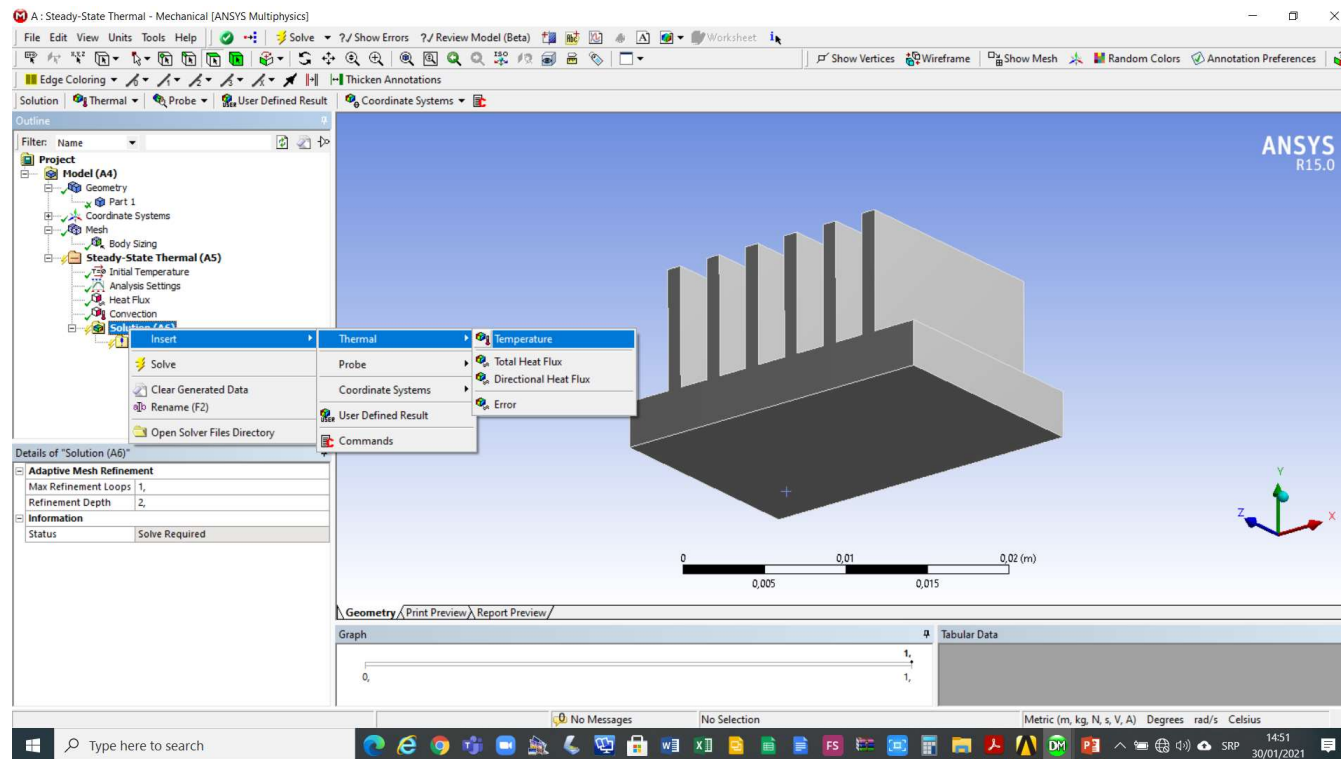
Toplotno opterećenje

Podešavanje temperature vazduha *Details of Convection*->*Definition*->*Ambient Temperature* na 28 °C



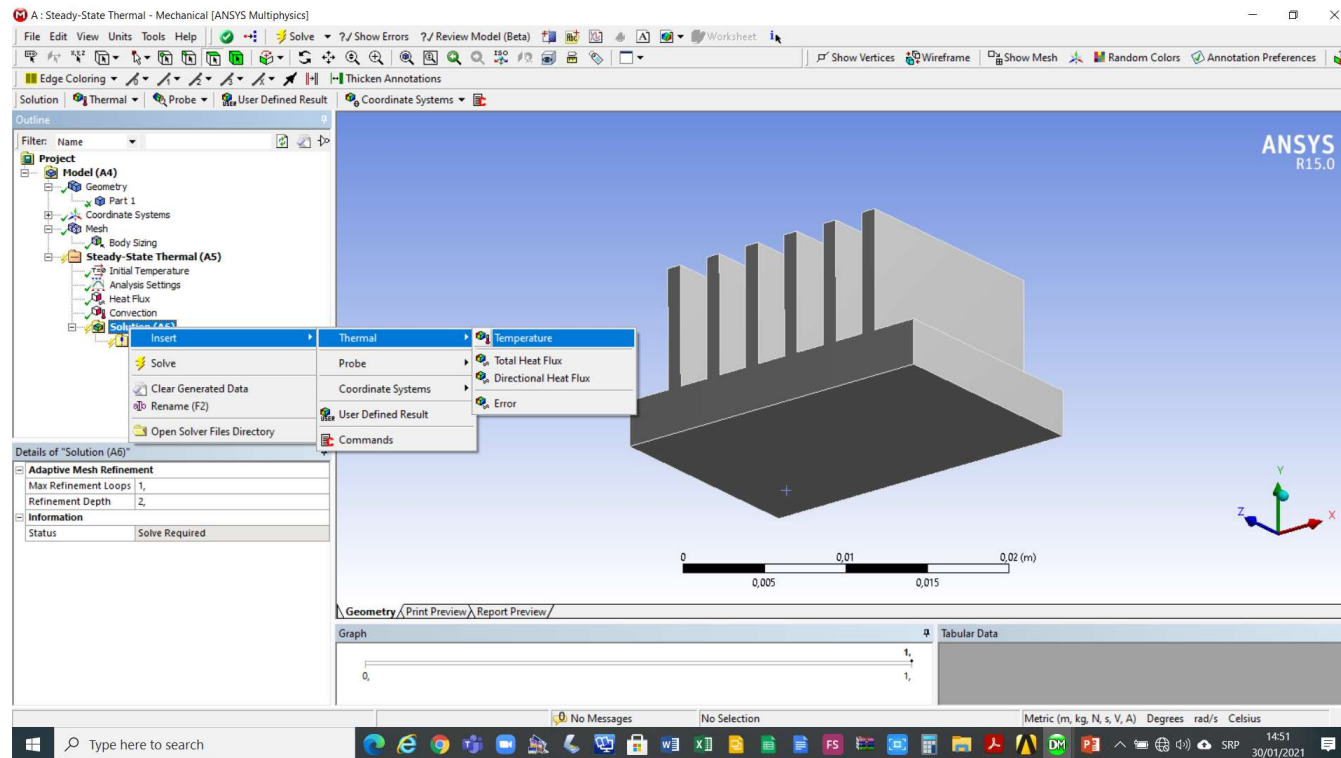
Toplotno opterećenje

Izabrati analizu koja se želi realizovati *Solution->Insert->Thermal->Temperature*



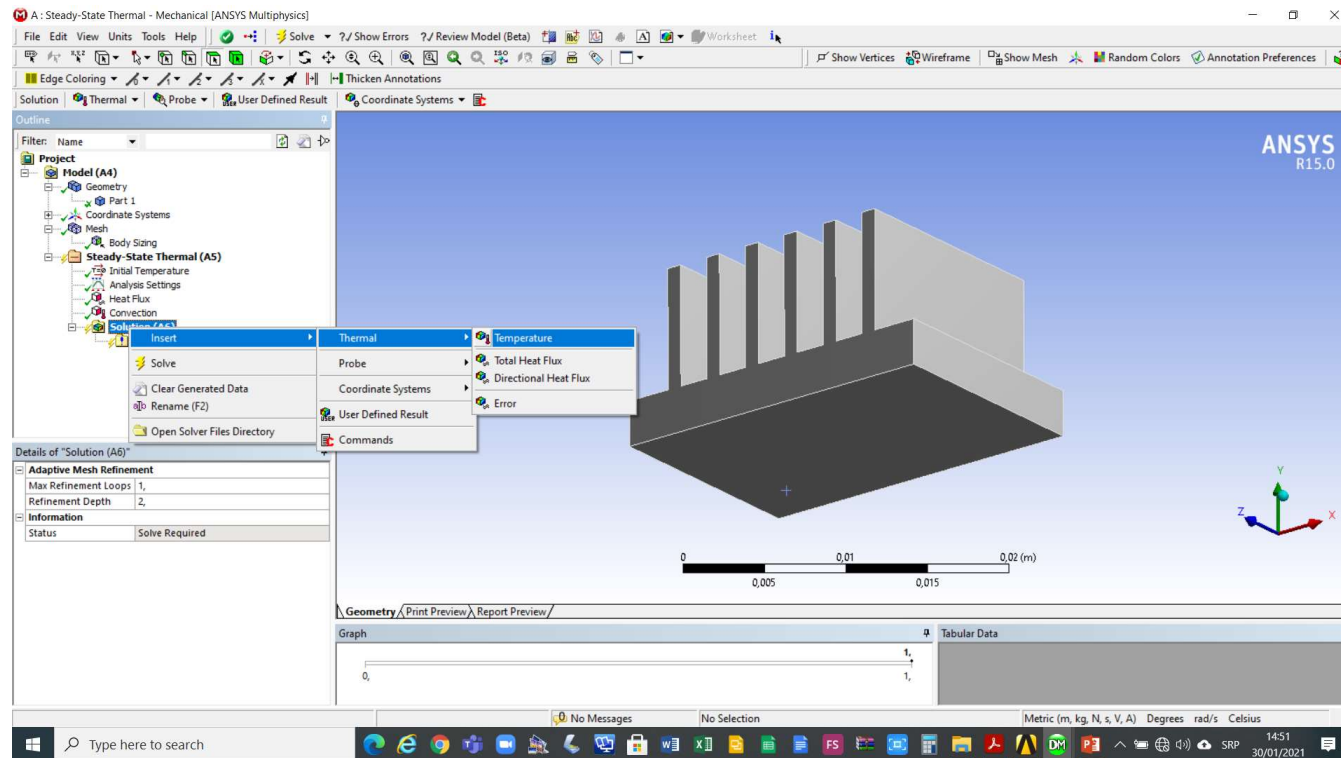
Toplotno opterećenje

Izabrati analizu koja se želi realizovati *Solution->Insert->Thermal->Total Heat Flux*



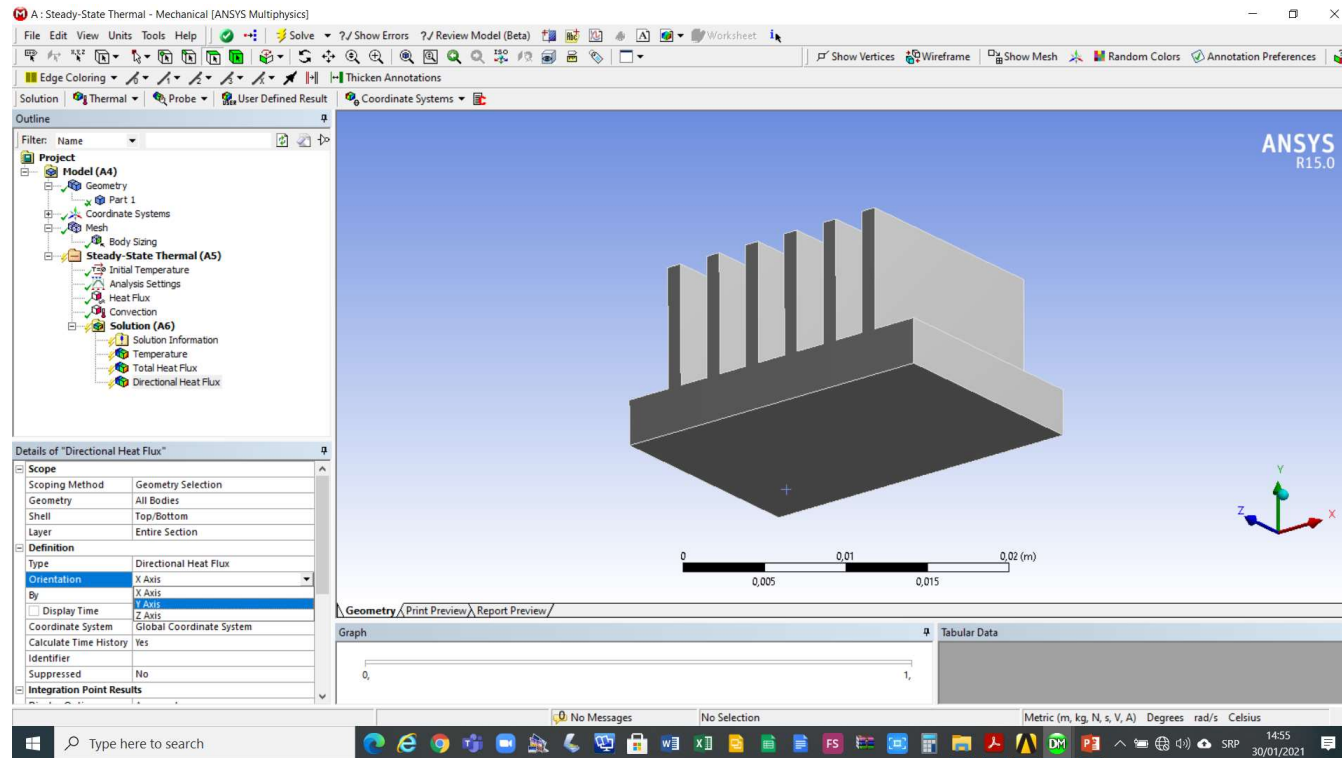
Toplotno opterećenje

Izabrati analizu koja se želi realizovati *Solution->Insert->Thermal->Directional Heat Flux*



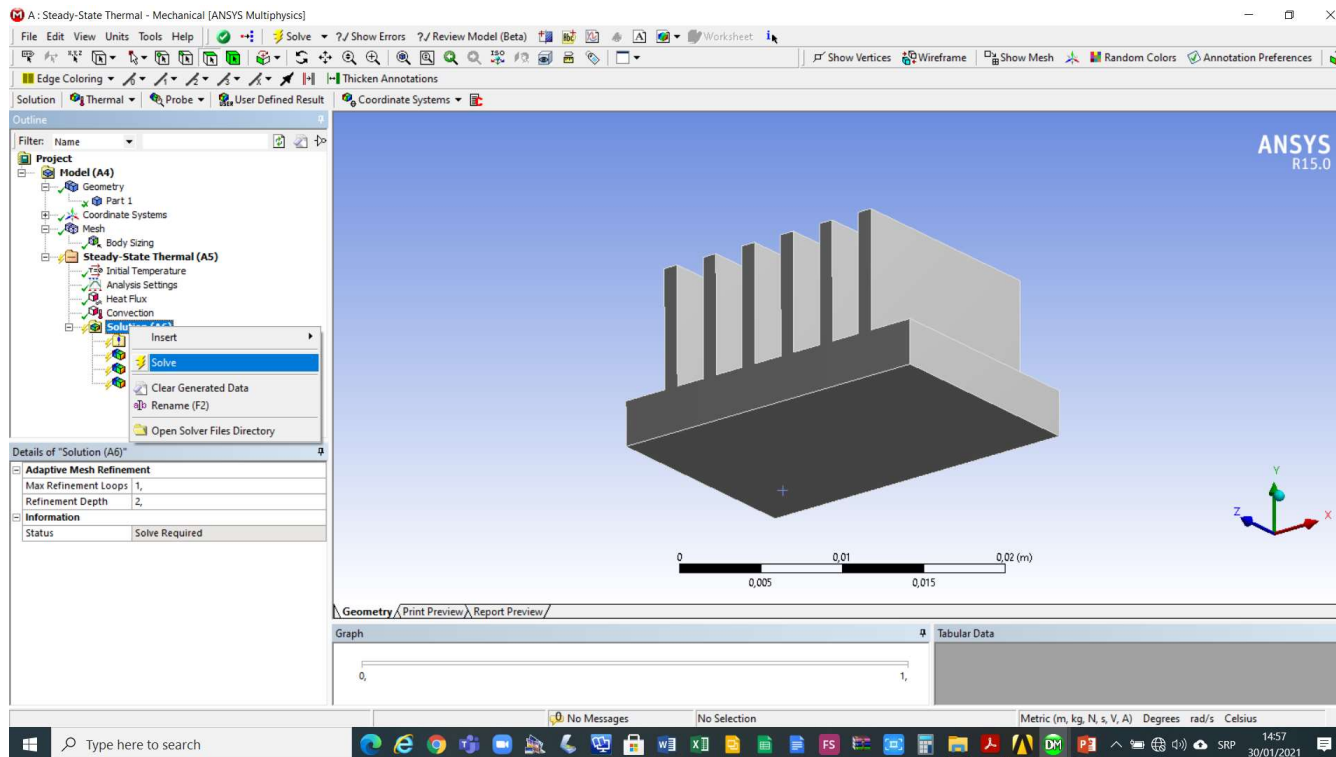
Toplotno opterećenje

Podesiti *Details of Directional Heat Flux*->*Definition*->*Orientation* na Y Axis



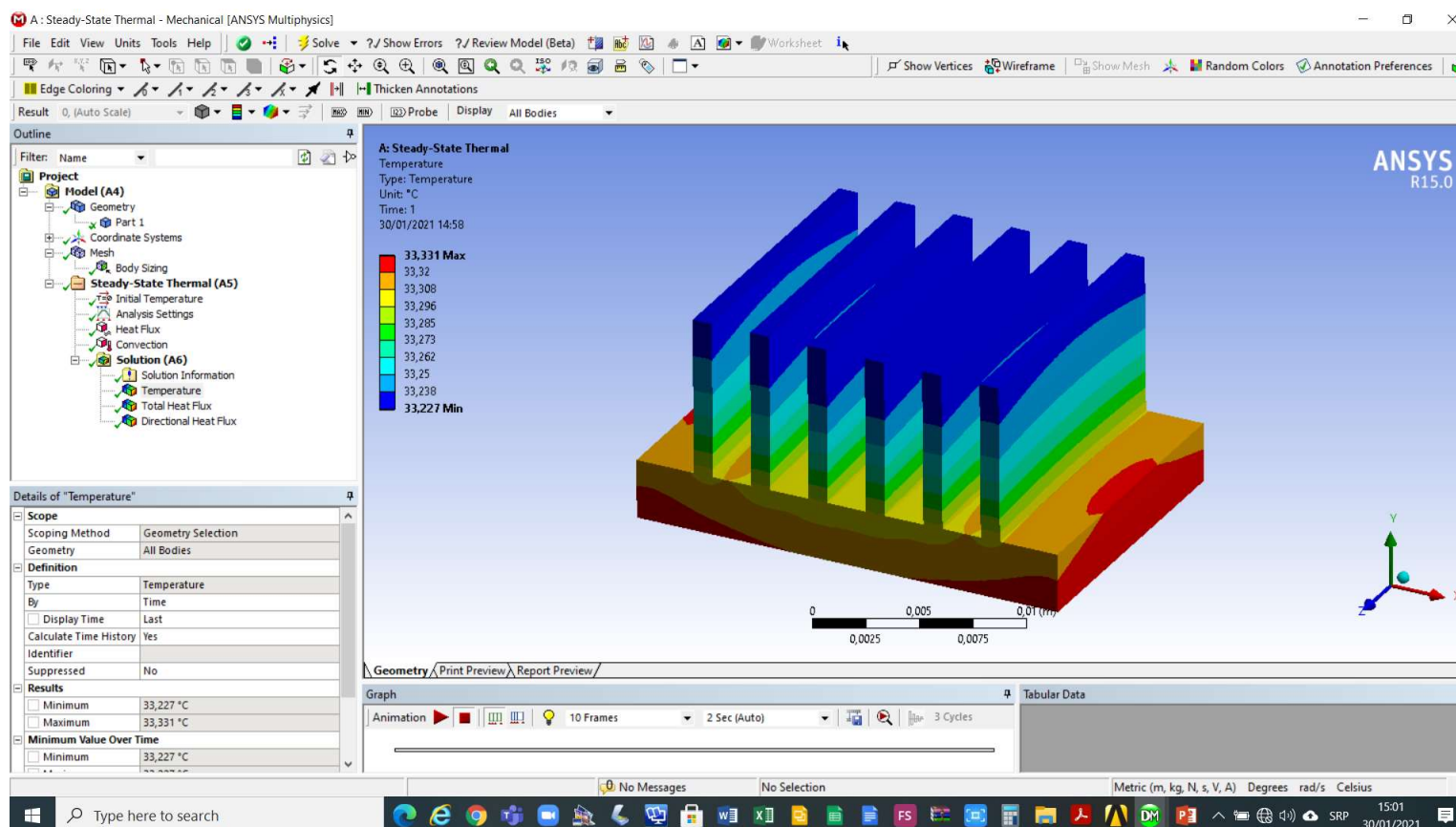
Toplotno opterećenje

Aktivirati izvršenje analize *Solution*->*Solve*



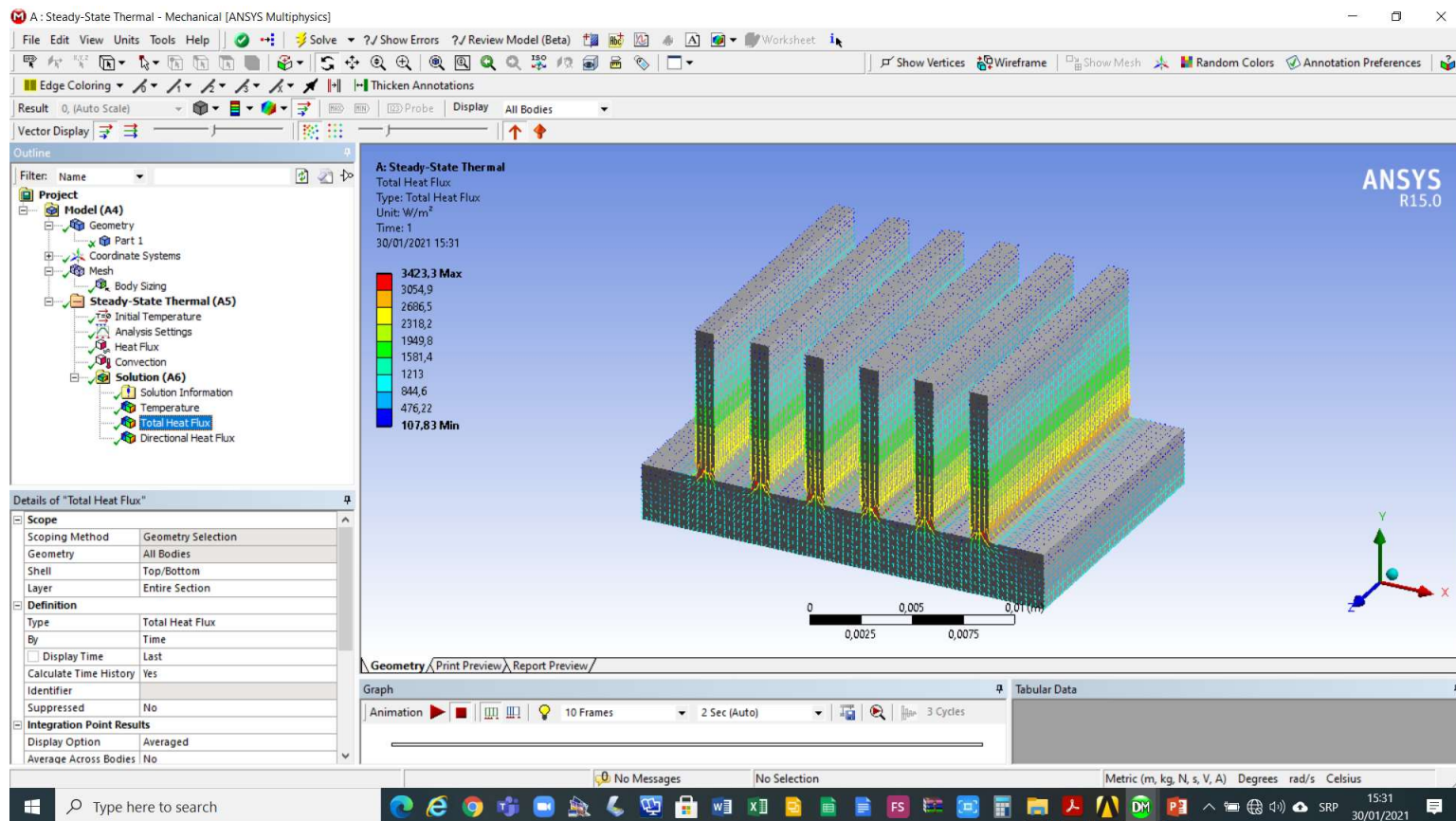
Toplotno opterećenje

Raspodjela temperature u ravnotežnom stanju



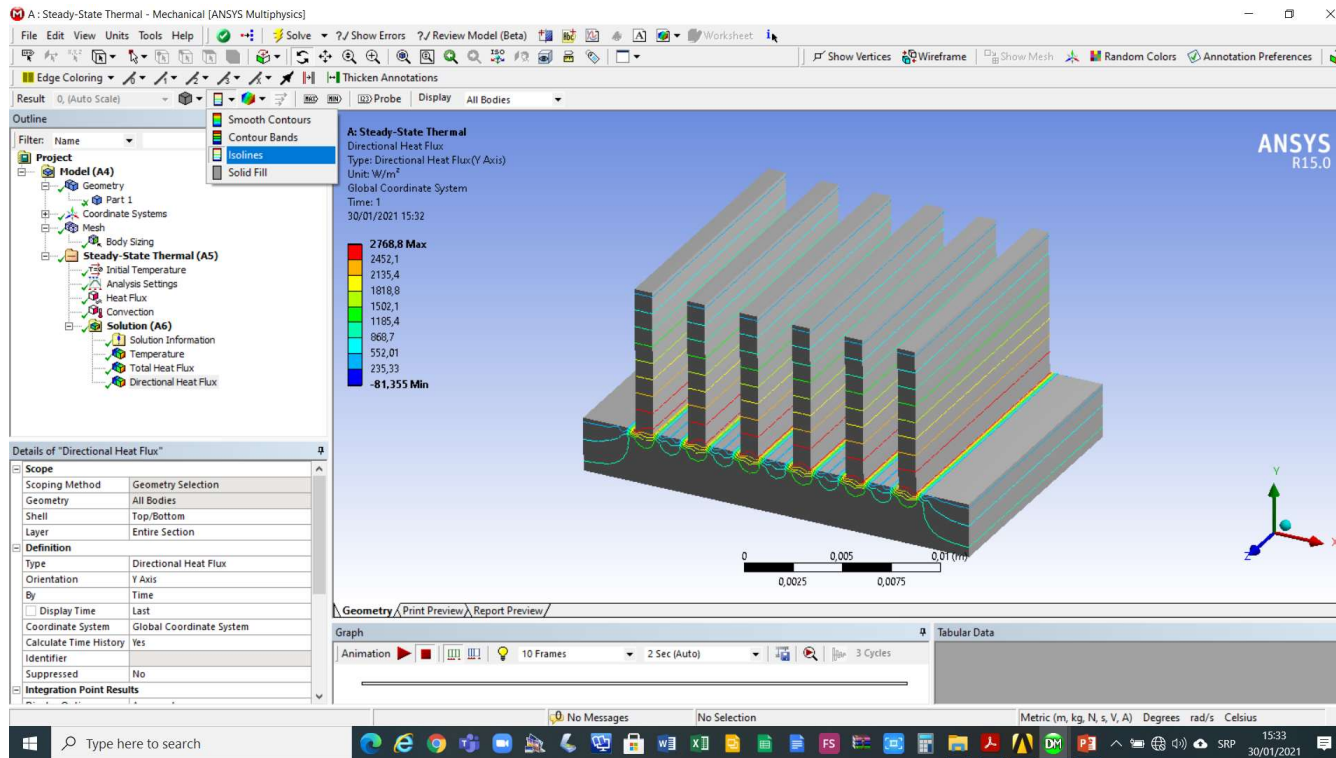
Toplotno opterećenje

Raspodjela toplotnog fluksa



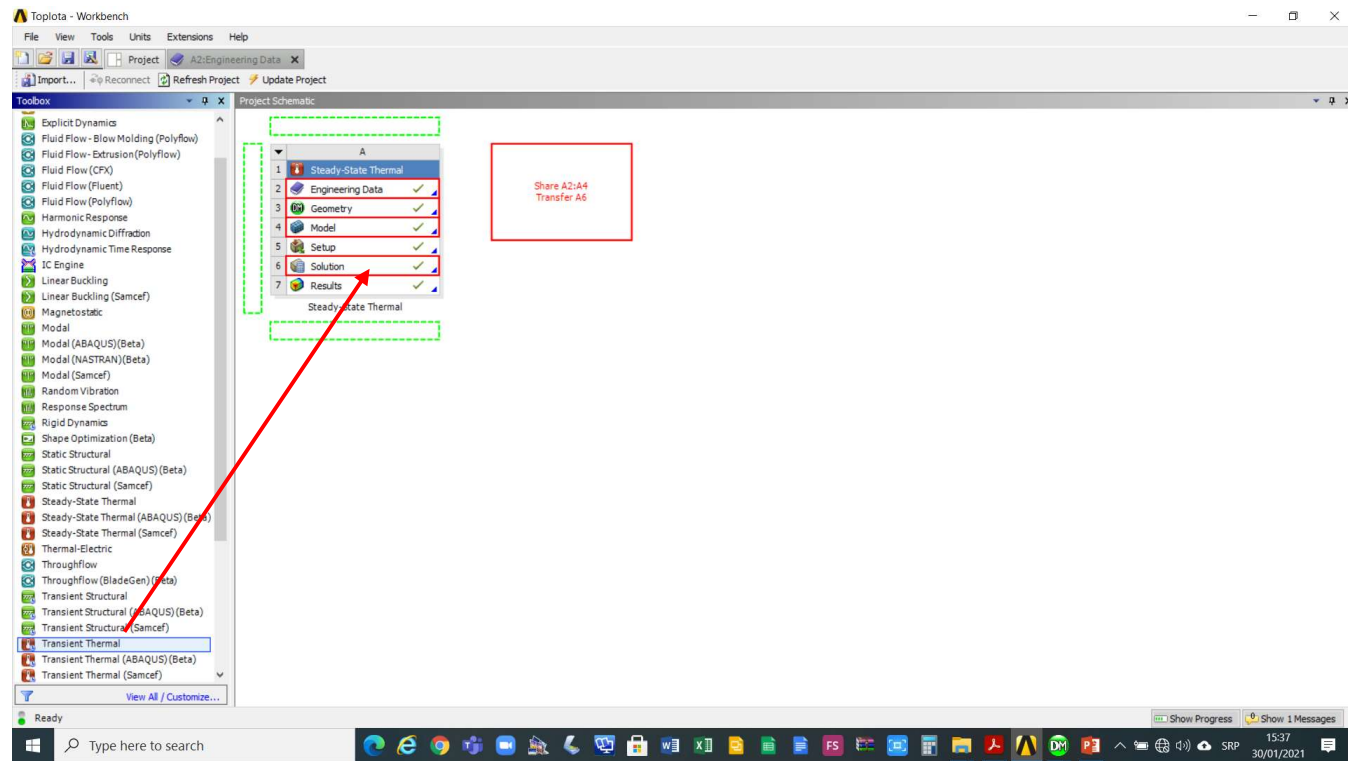
Toplotno opterećenje

Raspodjela toplotnog fluksa u pavcu normalnom na donju površinu



Toplotno opterećenje

Kreirati nestacionarnu termalnu analizu (*Transient Thermal*) povlačenjem na *Steady-State Thermal->Solution* stavku stacionarne analize



Toplotno opterećenje

Prethodni postupak omogućava potrebno povezivanje stacionarne i netacionarne termalne analize



Toplotno opterećenje

Aktivirati modul Engineering Data (*Engineering Data->Edit*) i dodati podatke o gustini i specifičnoj toploti

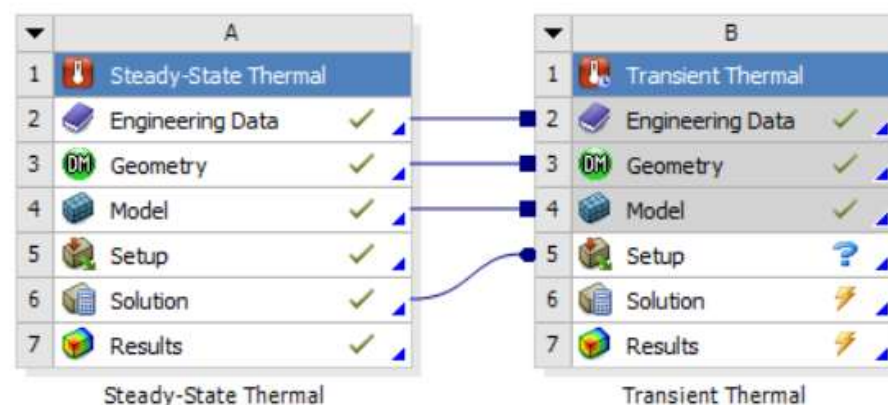
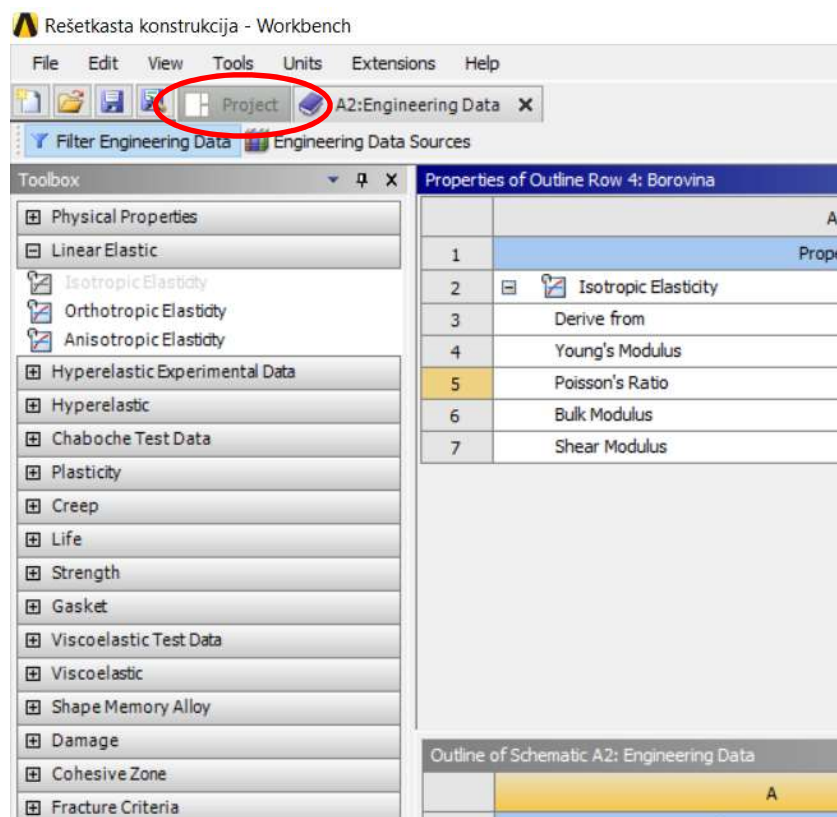
The screenshot displays the ANSYS Workbench Engineering Data interface. On the left, the 'Edit...' menu is open, showing options like Duplicate, Transfer Data From New, Update, Refresh, Rename, Properties, Quick Help, and Add Note. The main window shows the 'Outline of Schematic A2, B2: Engineering Data' with a table of materials. The 'Properties of Outline Row 4: Structural Steel' table is visible, showing the following data:

Property	Value	Unit
Density	2800	kg m ⁻³
Isotropic Thermal Conductivity	60,5	W m ⁻¹ C ⁻¹
Specific Heat	870	J kg ⁻¹ C ⁻¹

Below the table, a 'Chart of Properties Row 4: Specific Heat' is shown, plotting Specific Heat (J kg⁻¹ C⁻¹) on the y-axis (ranging from 0.5 to 1.3) against Temperature (C) on the x-axis (ranging from -1 to 1). A single data point is plotted at (0, 870).

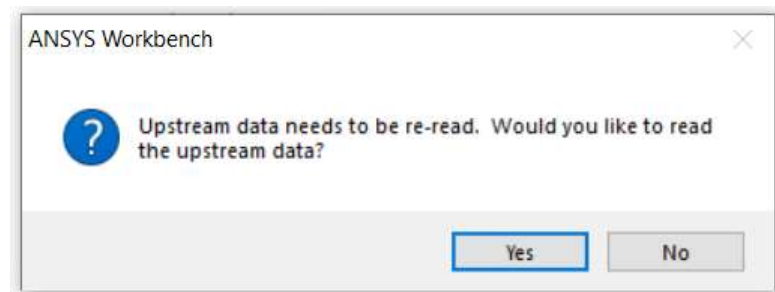
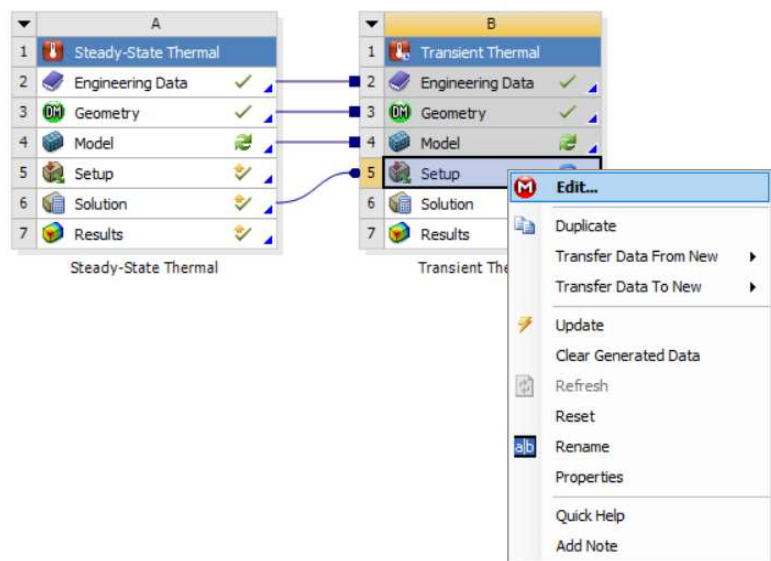
Toplotno opterećenje

Izabrati opciju *Project* za povratak na shemu projekta



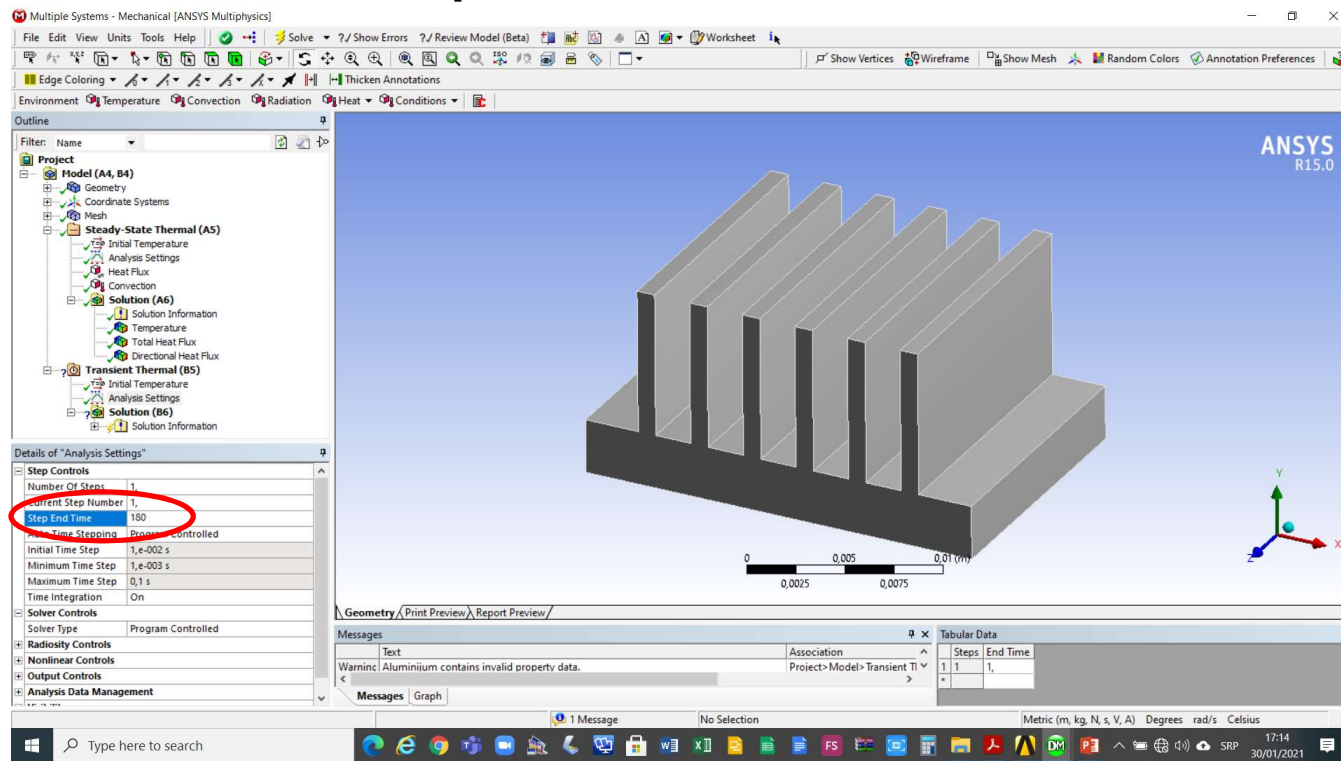
Toplotno opterećenje

Aktivirati modul Multiple Systems (*Transient Thermal*->*Setup*->*Edit*) i odgovoriti sa *Yes* kako bi se učitali novounešeni podaci



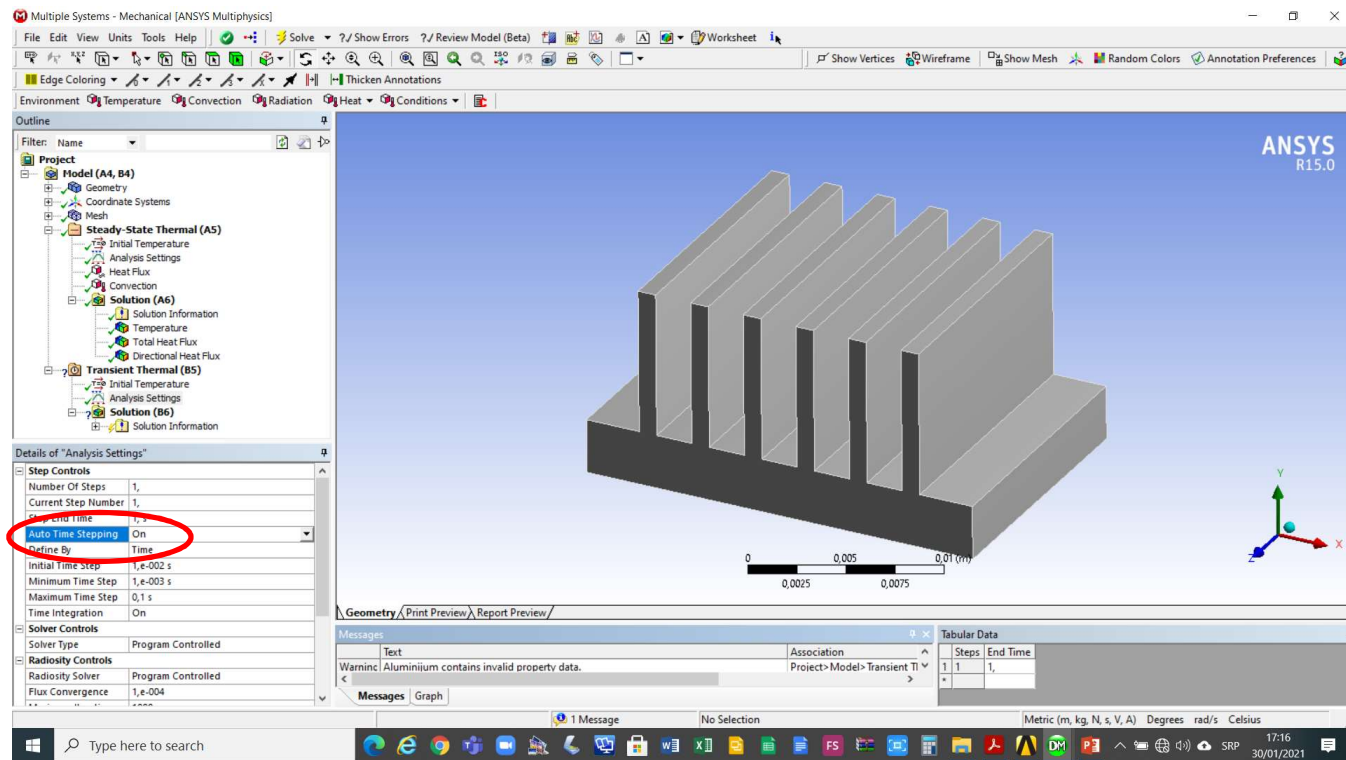
Toplotno opterećenje

Podesiti vrijeme trajanja analize nestacionarnog prenosa toplote *Details of Analysis Settings->Step Control->Step End Time* na 180 s



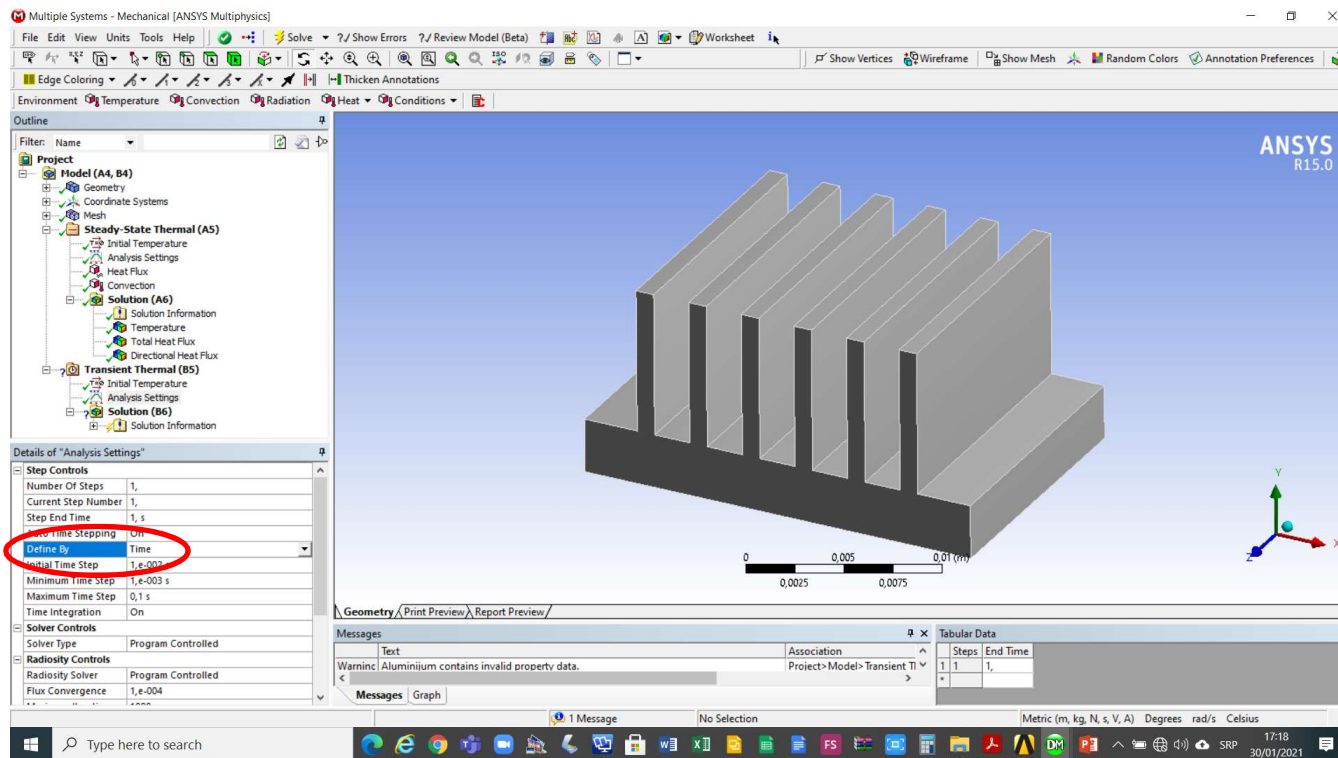
Toplotno opterećenje

Podesiti način izbora vremenskog koraka *Details of Analysis Settings*->*Step Control*->*Auto Time Stepping* na On



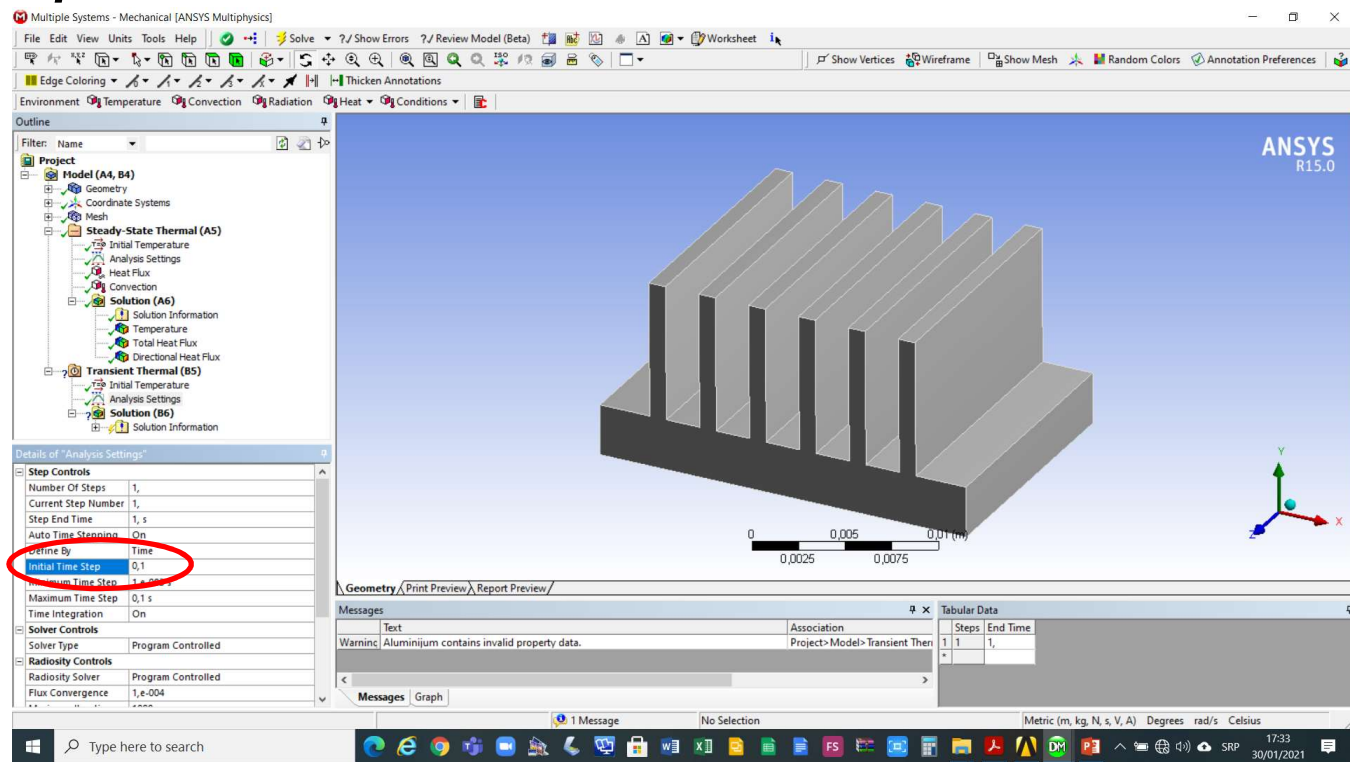
Toplotno opterećenje

Izabrati tip koraka analize *Details of Analysis Settings*
Settings->*Step Control*->*Define By* na Time



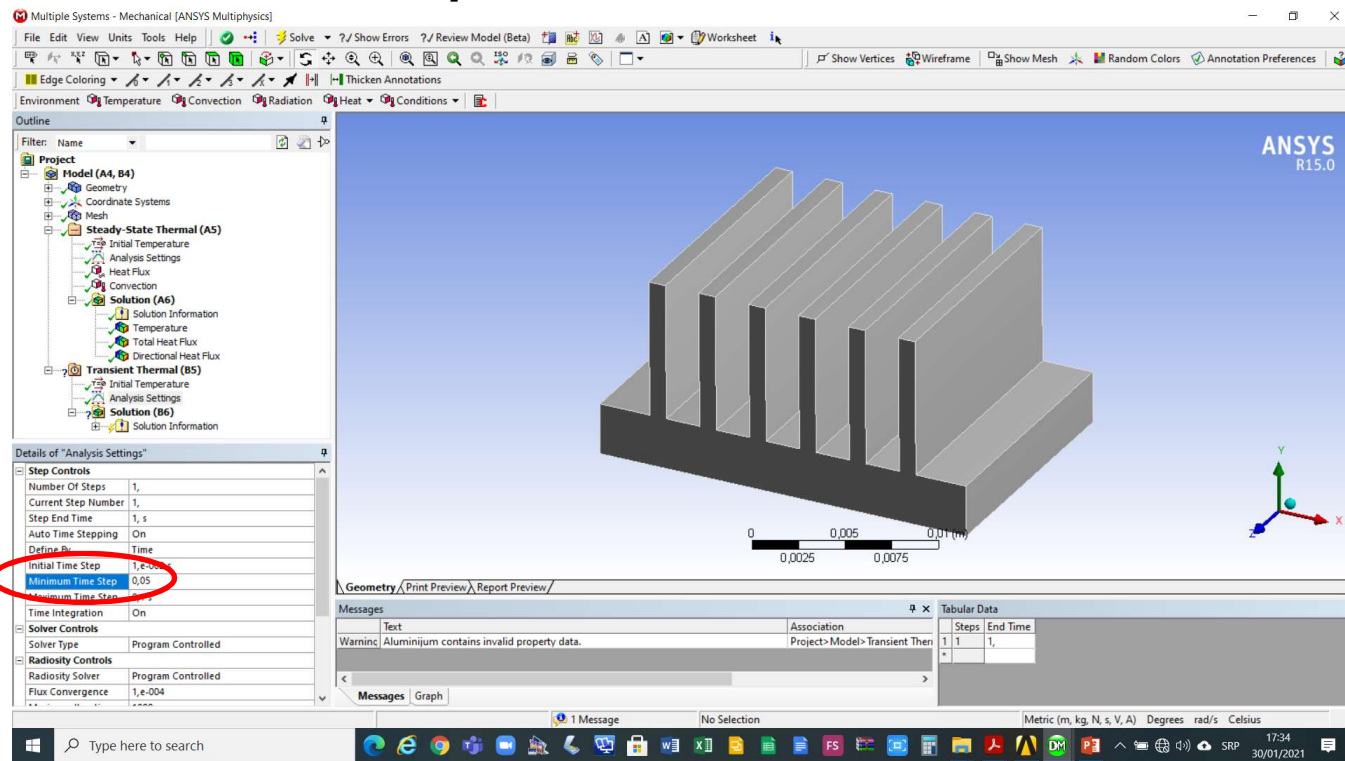
Toplotno opterećenje

Podesiti parametre vremenskog koraka analize
Details of Analysis Settings->*Step Control*->*Initial Time Step* na 0.1 s



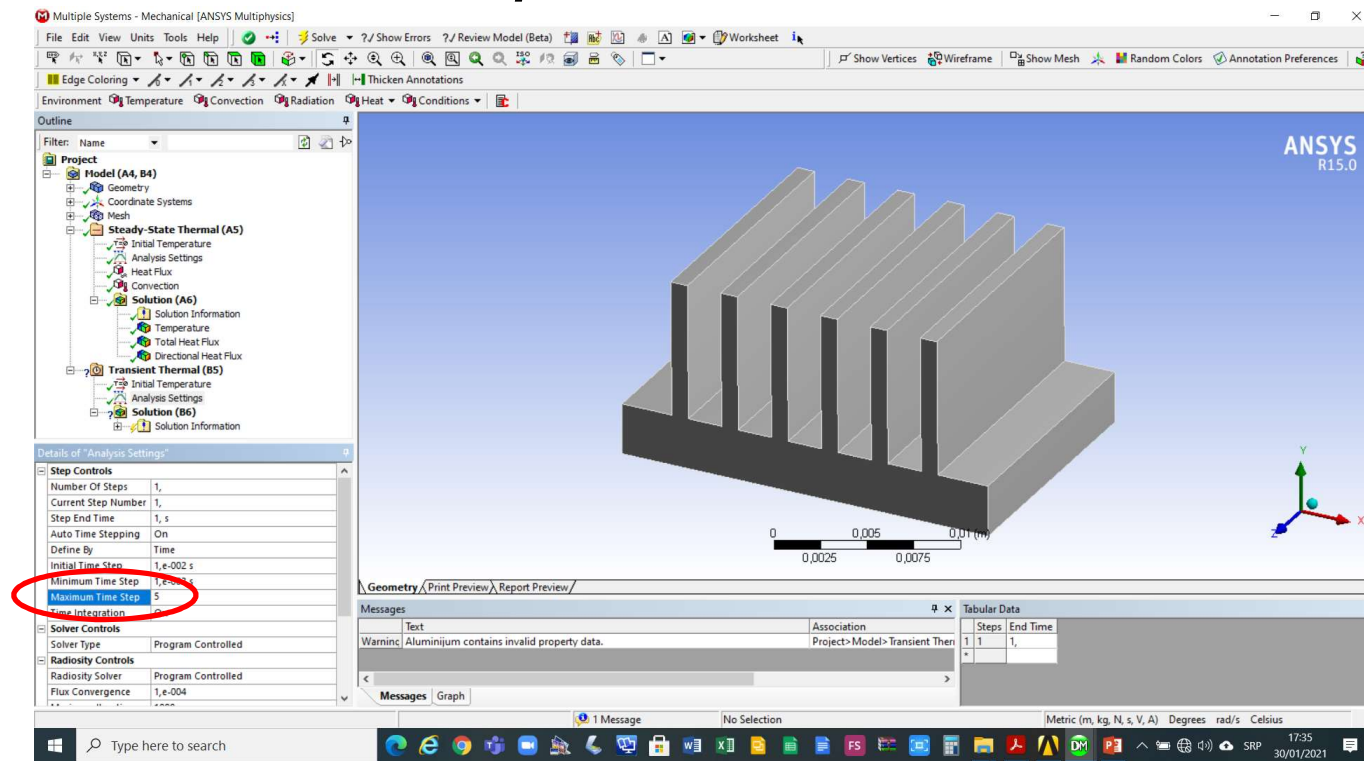
Toplotno opterećenje

Podesiti parametre vremenskog koraka analize
Details of Analysis Settings->Step Control->Minimum Time Step na 0.05 s



Toplotno opterećenje

Podesiti parametre vremenskog koraka analize
*Details of Analysis Settings->Step Control-
>Maximum Time Step na 5 s*

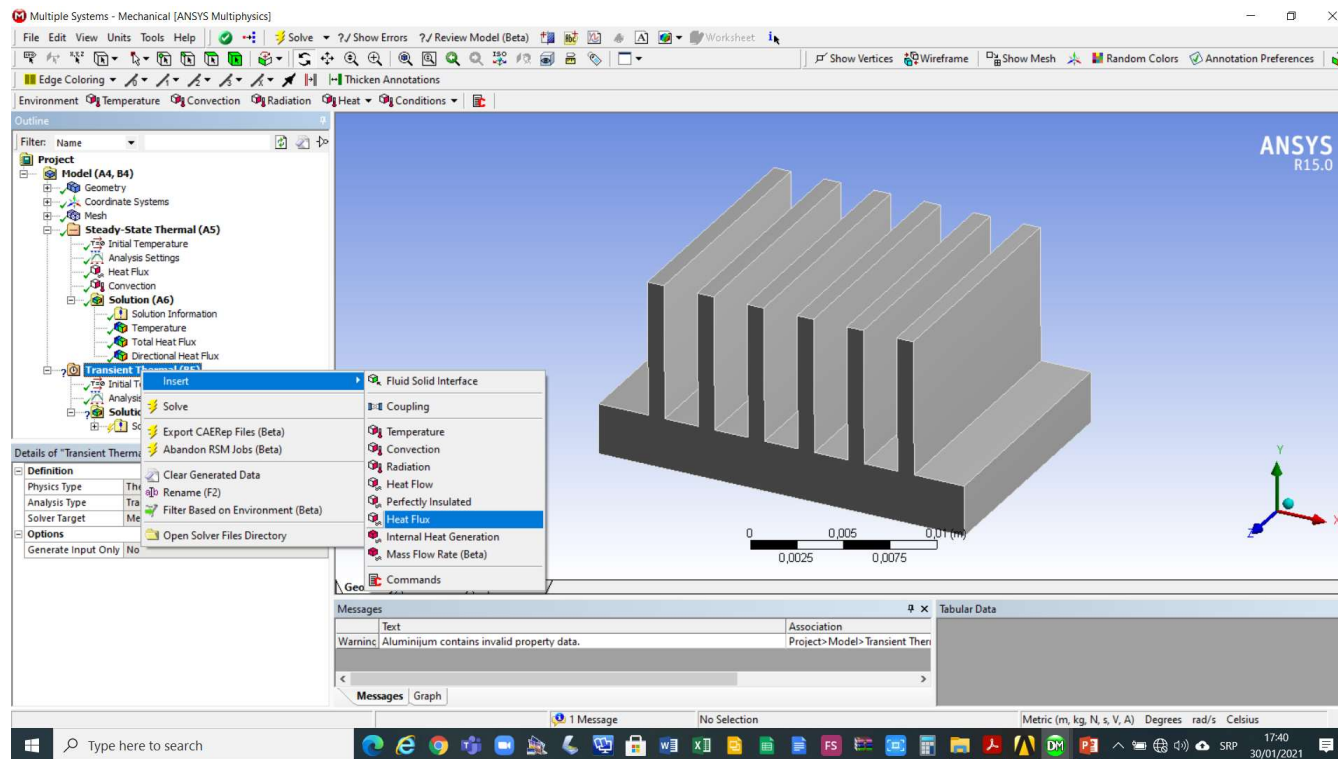


Toplotno opterećenje

Dovoljno mali vremenski korak povećava tačnost analize i omogućava dovoljan broj koraka koji rezultira simulacijom sa finim prelaskom sa između koraka analize

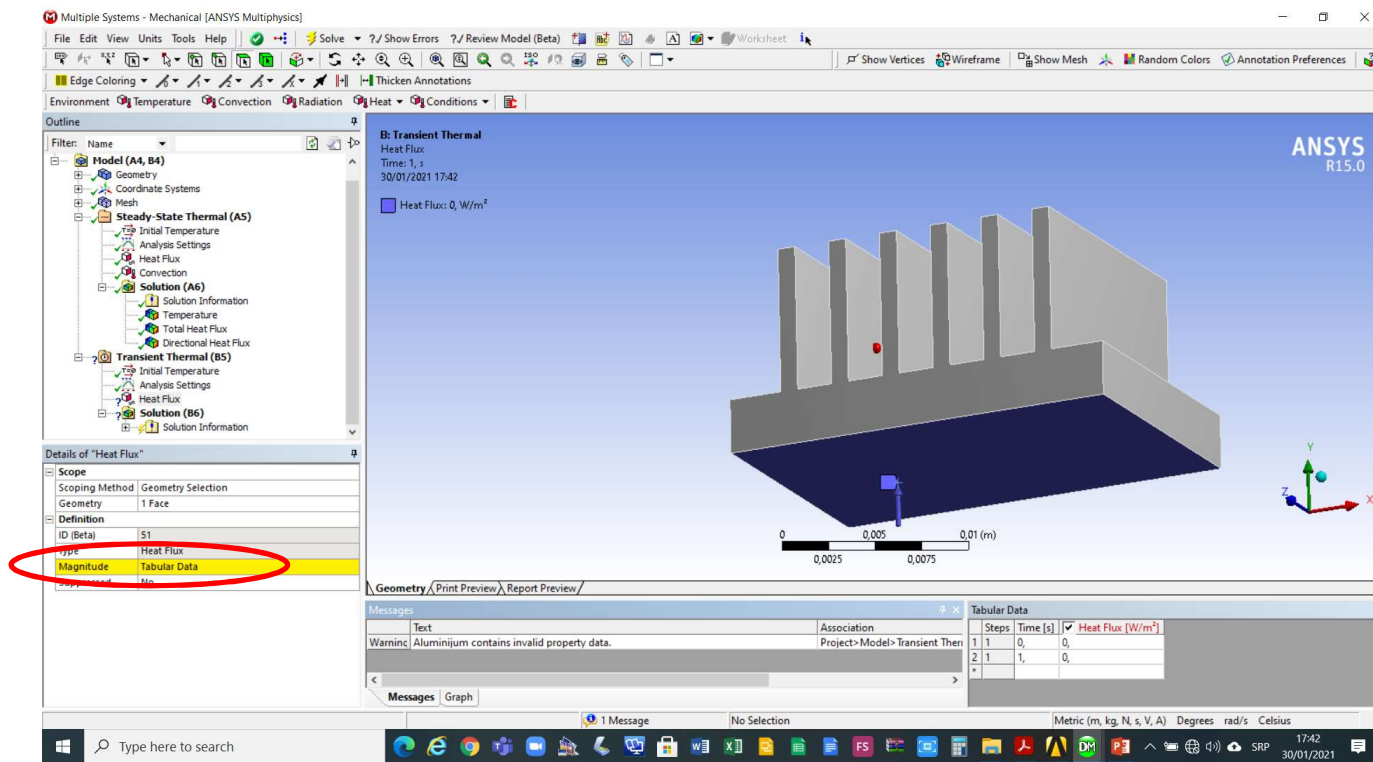
Toplotno opterećenje

Dodati toplotni fluks u model *Transient Thermal*
>Insert->Heat Flux



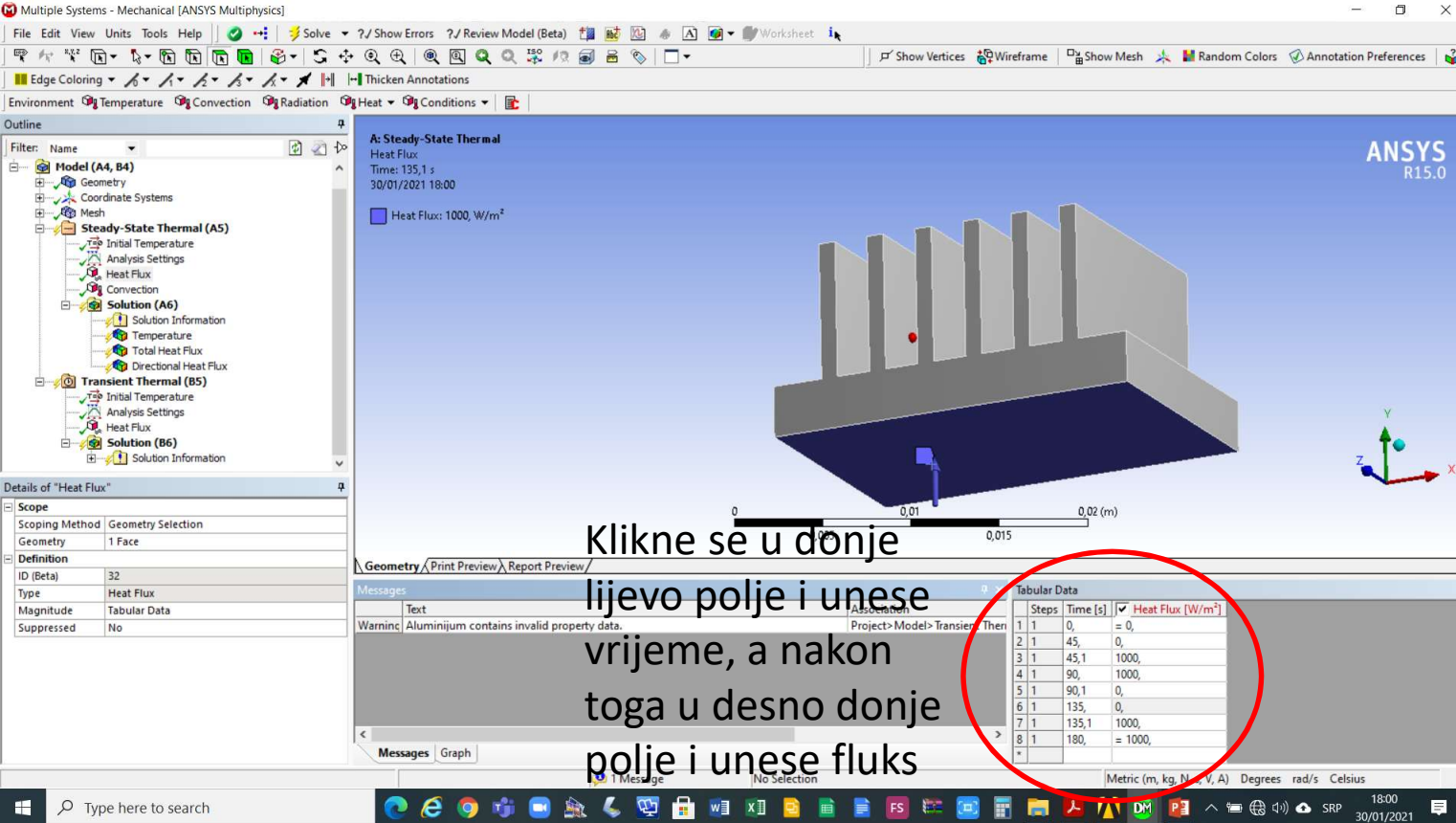
Toplotno opterećenje

Podešavanje veličine toplotnog fluksa *Details of Heat Flux*->*Definition*->*Magnitude* koji djeluje po donjoj površini na Tabular Data



Toplotno opterećenje

Unijeti tabelarne podatke o toplotnom fluksu kao na slici

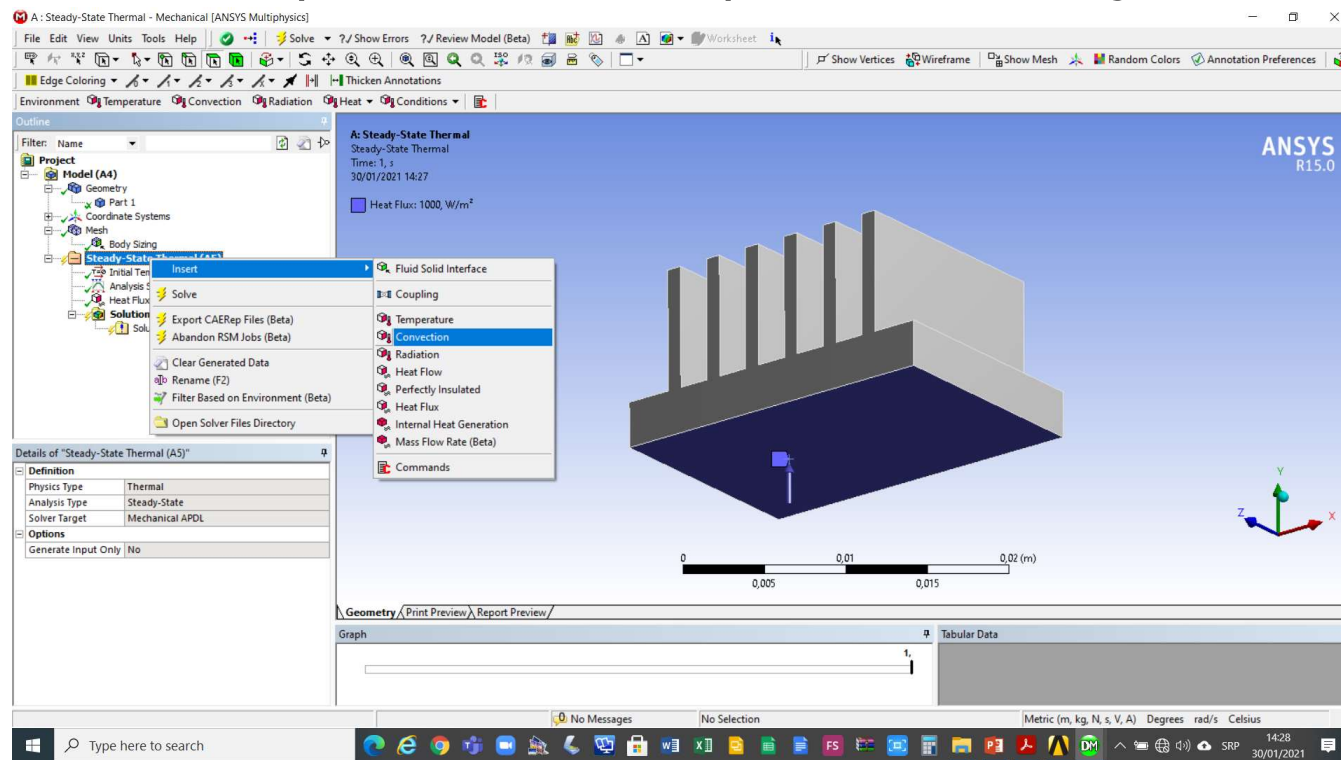


Klikne se u donje lijevo polje i unese vrijeme, a nakon toga u desno donje polje i unese fluks

Steps	Time [s]	Heat Flux [W/m ²]
1	0,	= 0,
2	45,	0,
3	45,1	1000,
4	90,	1000,
5	90,1	0,
6	135,	0,
7	135,1	1000,
8	180,	= 1000,

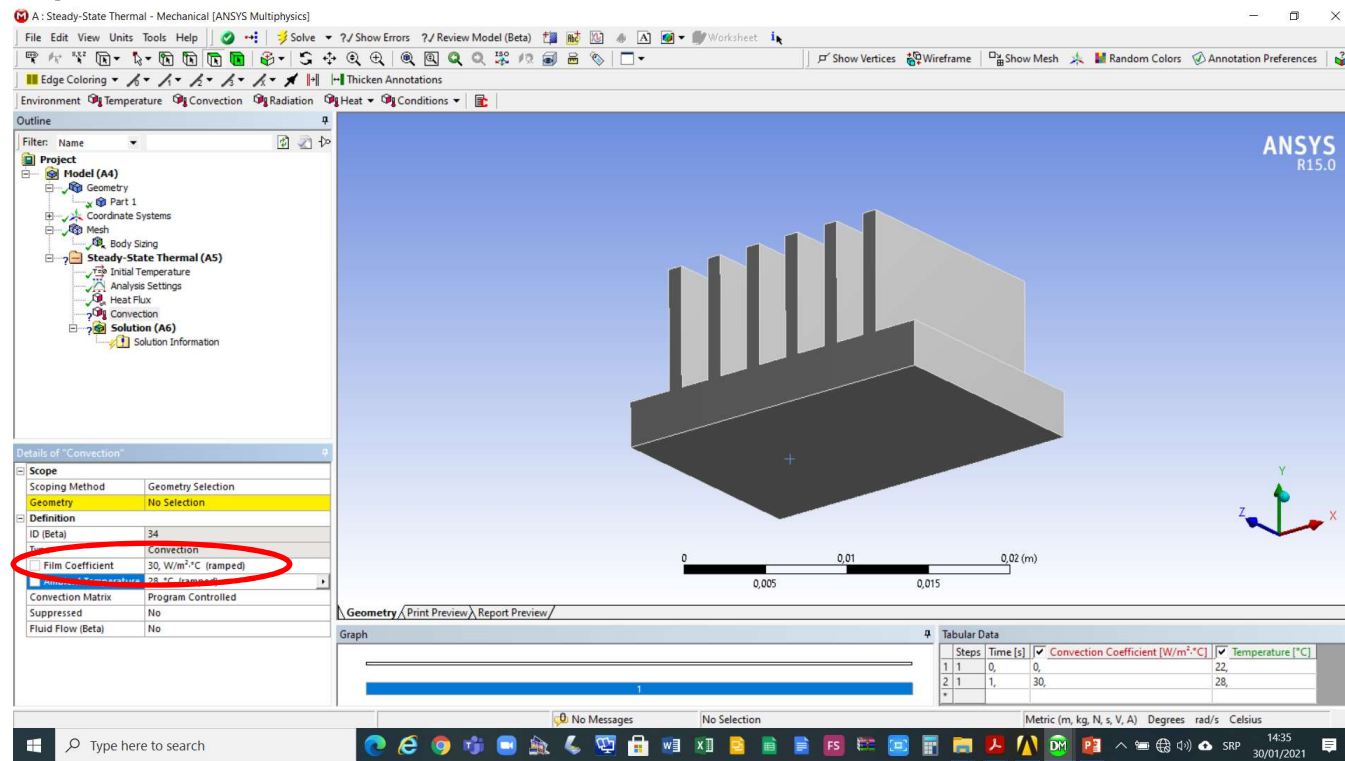
Toplotno opterećenje

Dodati konvektivni prenos toplote u model *Transient Thermal*->*Insert*->*Convection* koji djeluje na sve (29 površina), osim donje površine



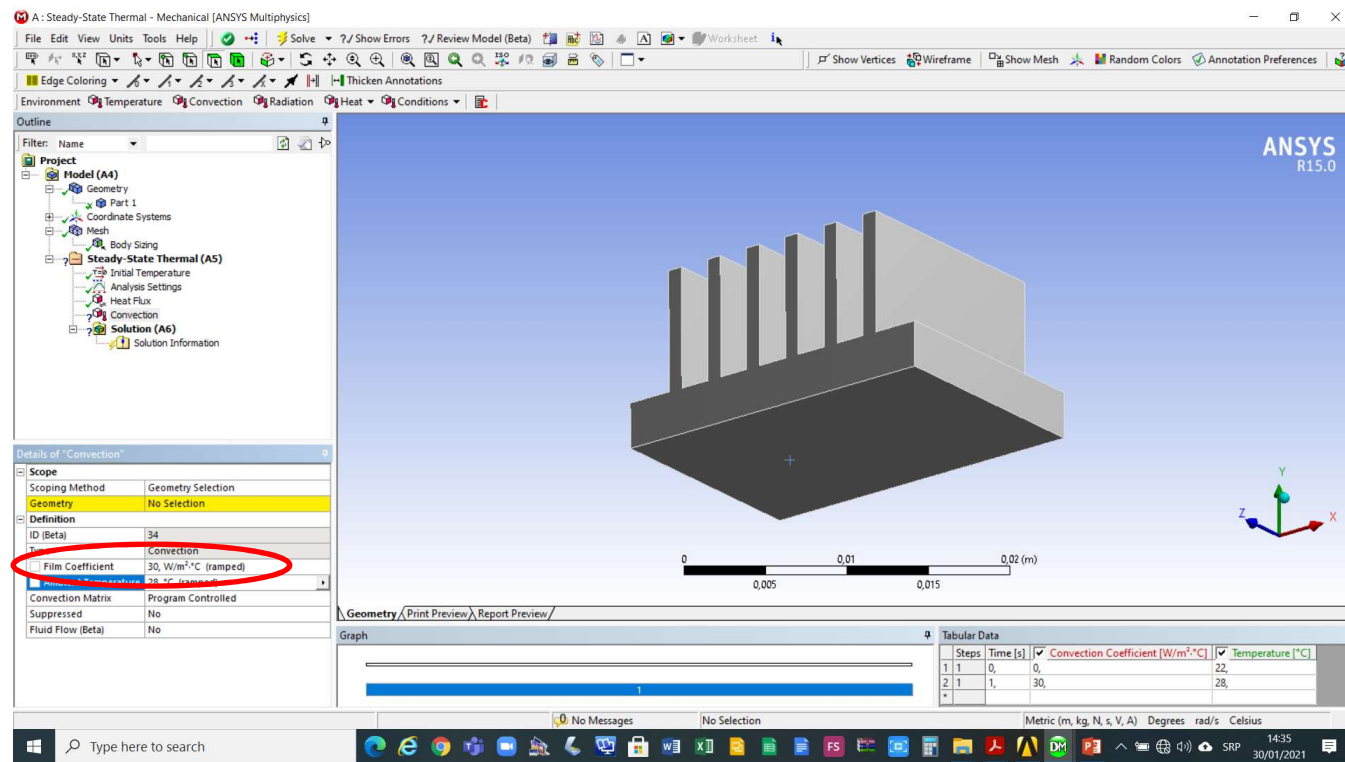
Toplotno opterećenje

Podešavanje koeficijenta prenosa toplote *Details of Convection*->*Definition*->*Film Coefficient* na $30 \text{ W}/(\text{m}^2 \cdot ^\circ\text{C})$



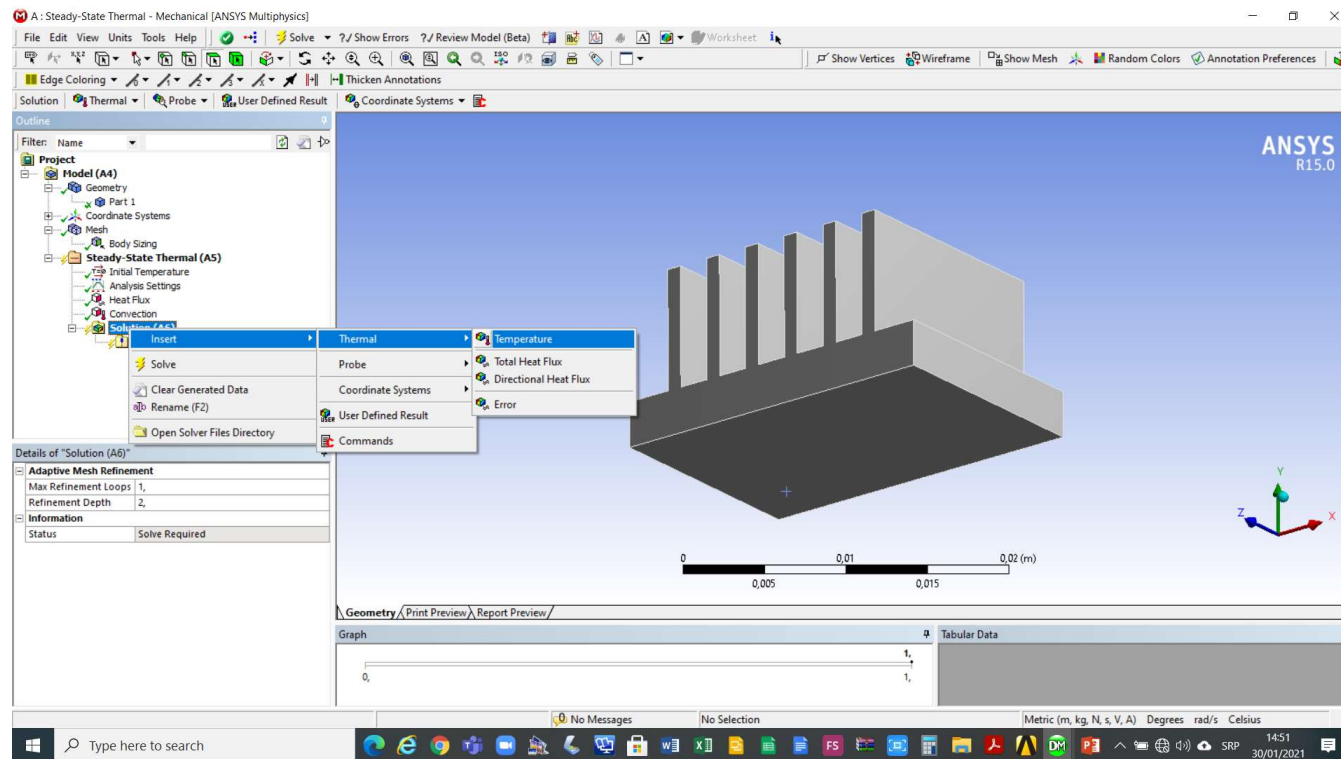
Toplotno opterećenje

Podešavanje temperature vazduha *Details of Convection*->*Definition*->*Ambient Temperature* na 28 °C



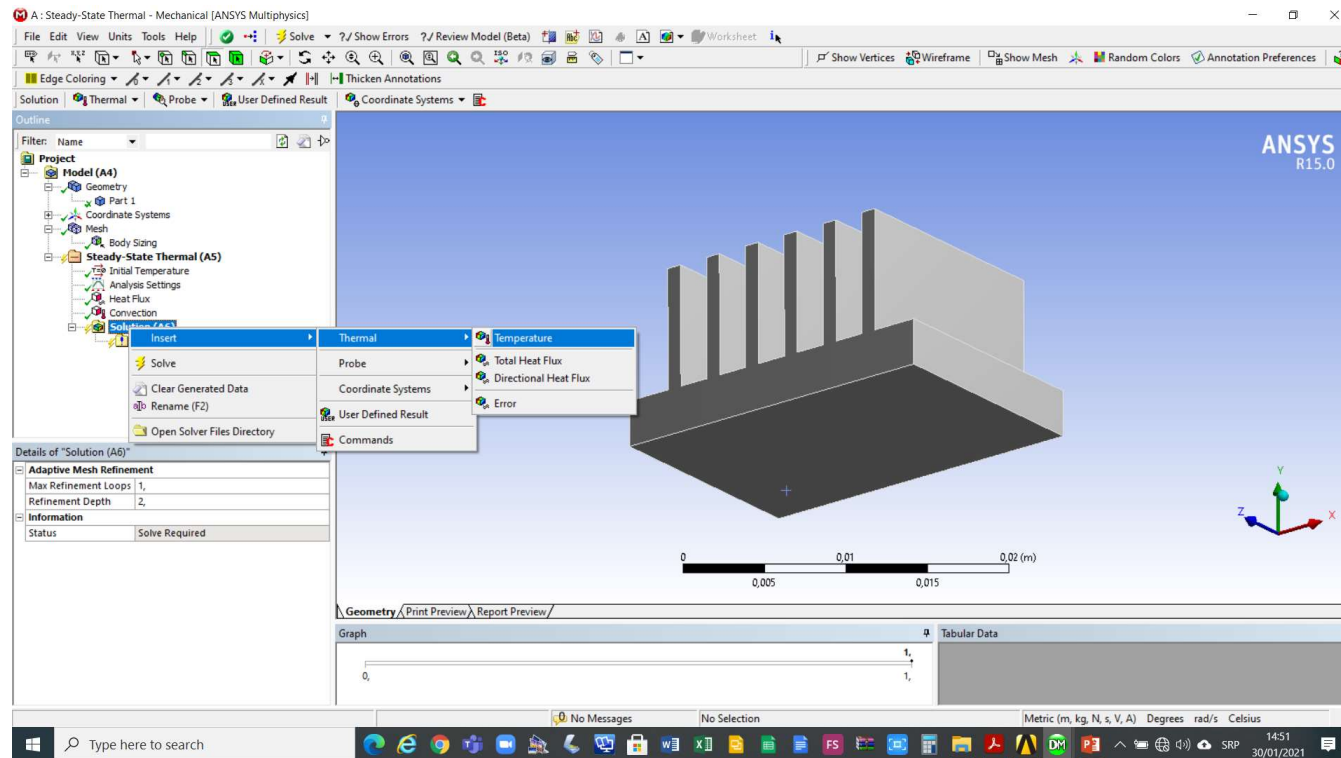
Toplotno opterećenje

Izabrati analizu koja se želi realizovati *Solution->Insert->Thermal->Temperature*



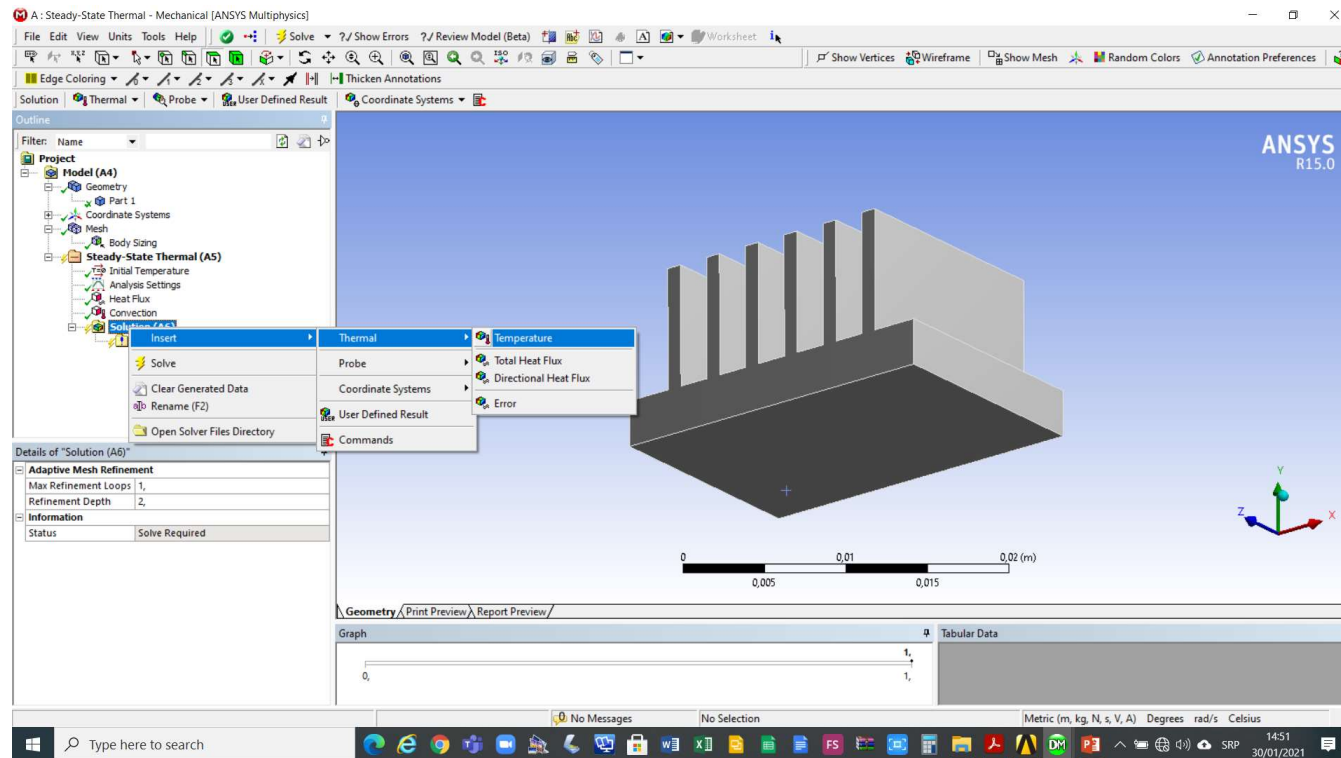
Toplotno opterećenje

Izabrati analizu koja se želi realizovati *Solution->Insert->Thermal->Total Heat Flux*



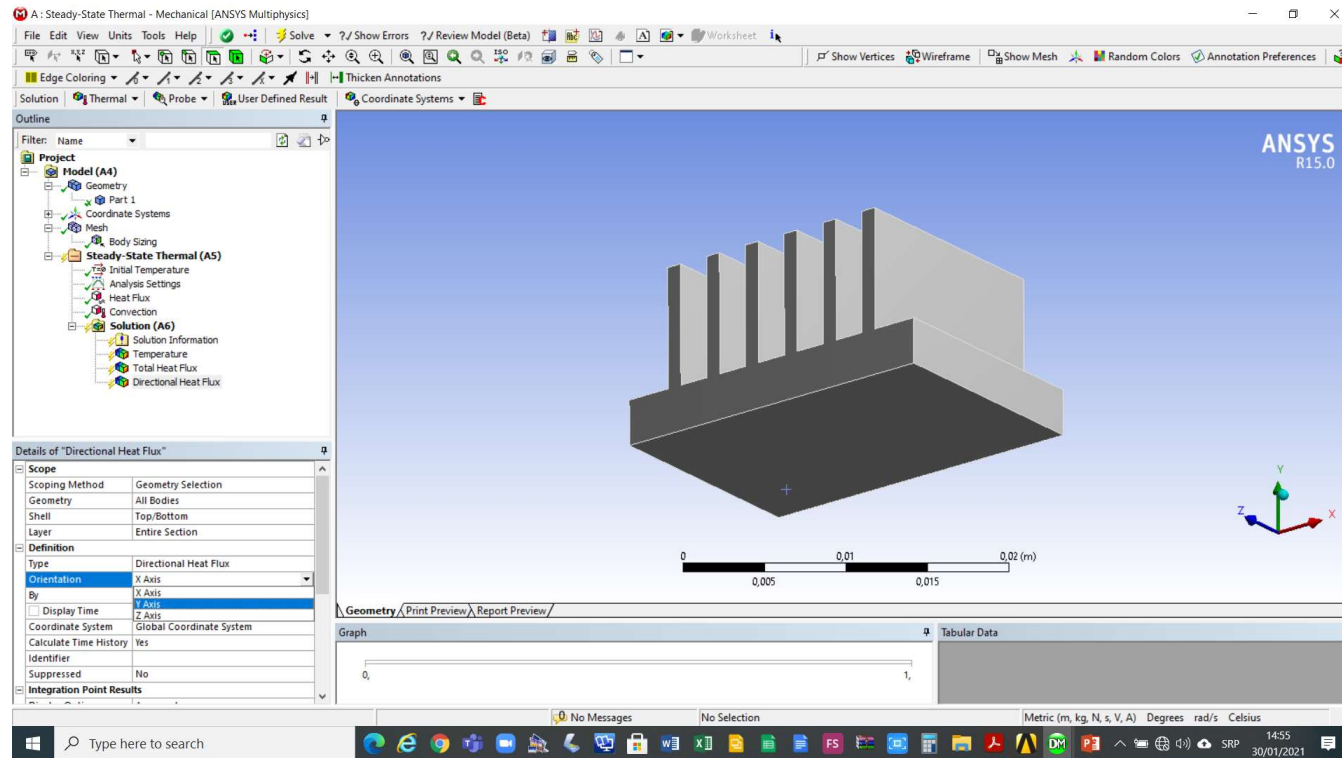
Toplotno opterećenje

Izabrati analizu koja se želi realizovati *Solution->Insert->Thermal->Directional Heat Flux*



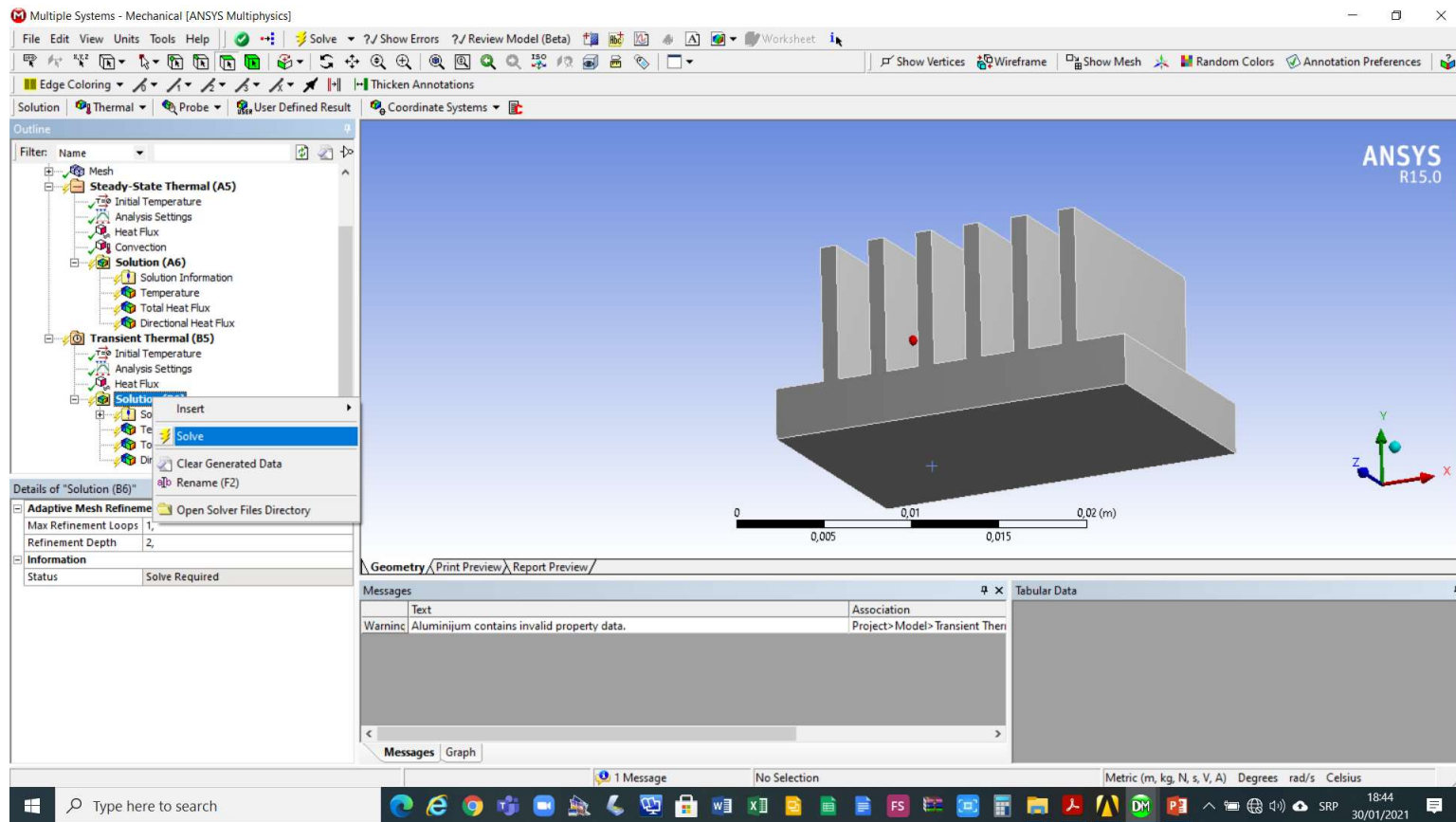
Toplotno opterećenje

Podesiti *Details of Directional Heat Flux*->*Definition*->*Orientation* na Y Axis



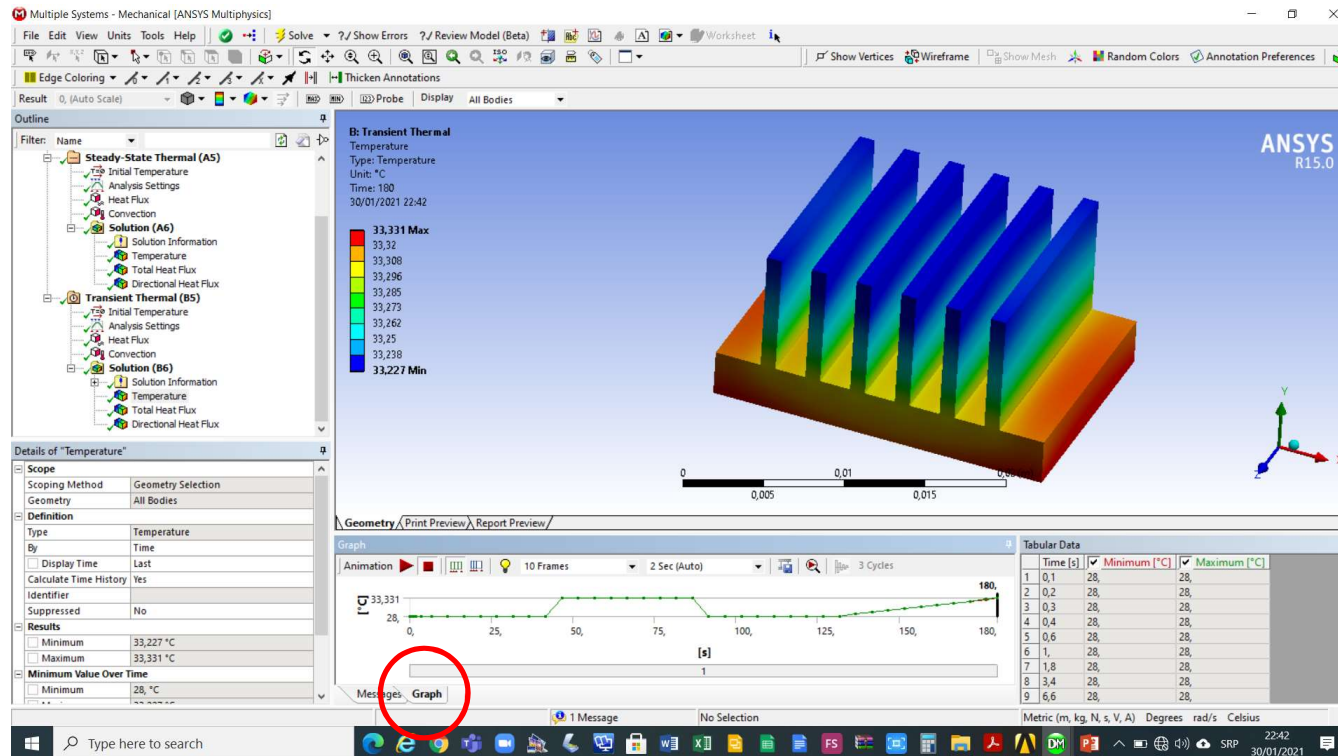
Toplotno opterećenje

Aktivirati izvršenje analize *Solution*->*Solve*



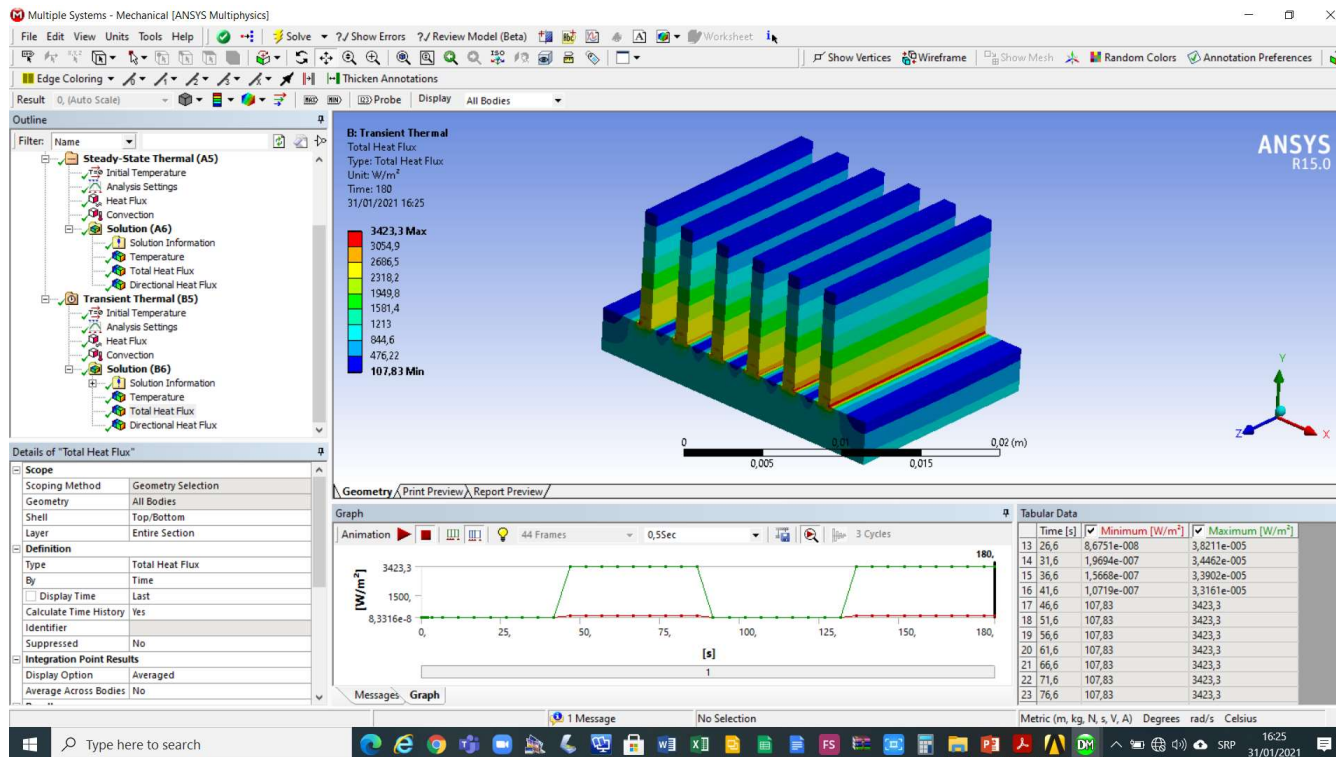
Toplotno opterećenje

Simulacija promjene raspodjele temperature
opcija *Graph*



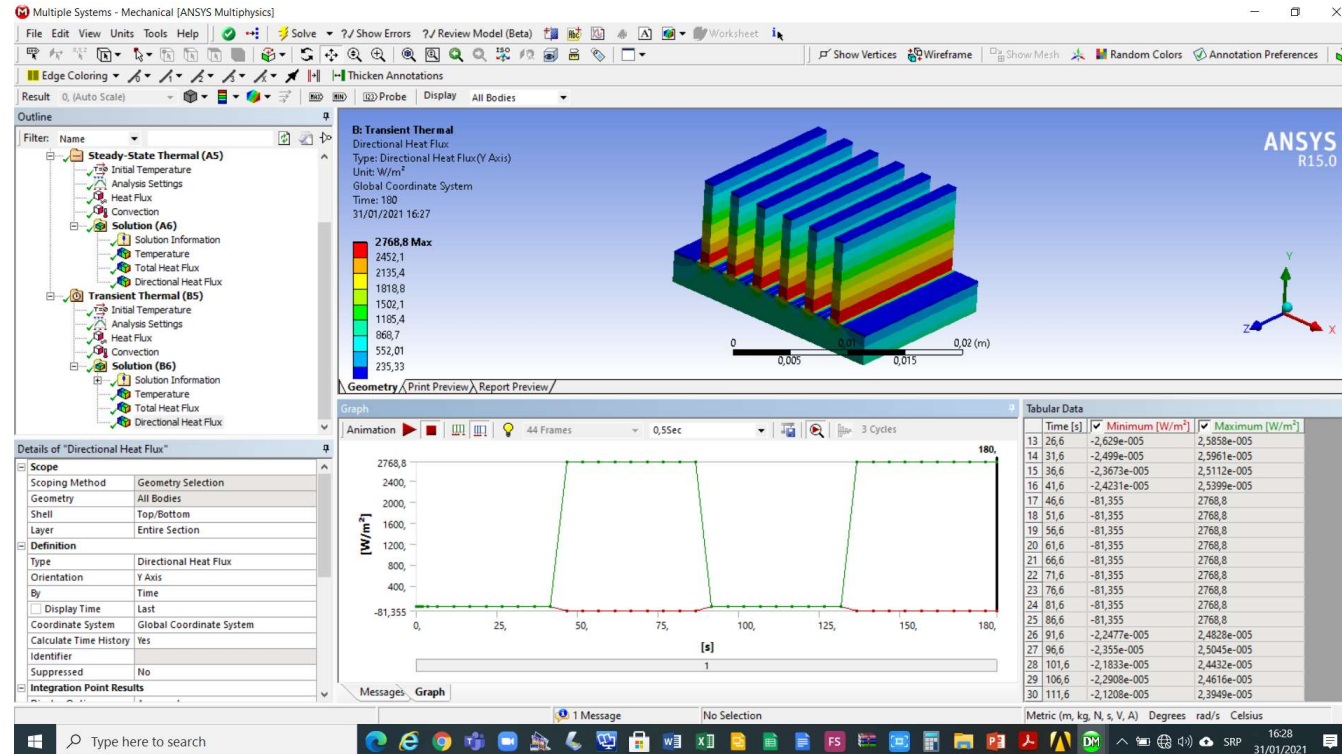
Toplotno opterećenje

Simulacija promjene raspodjele toplotnog fluksa opcija *Graph*



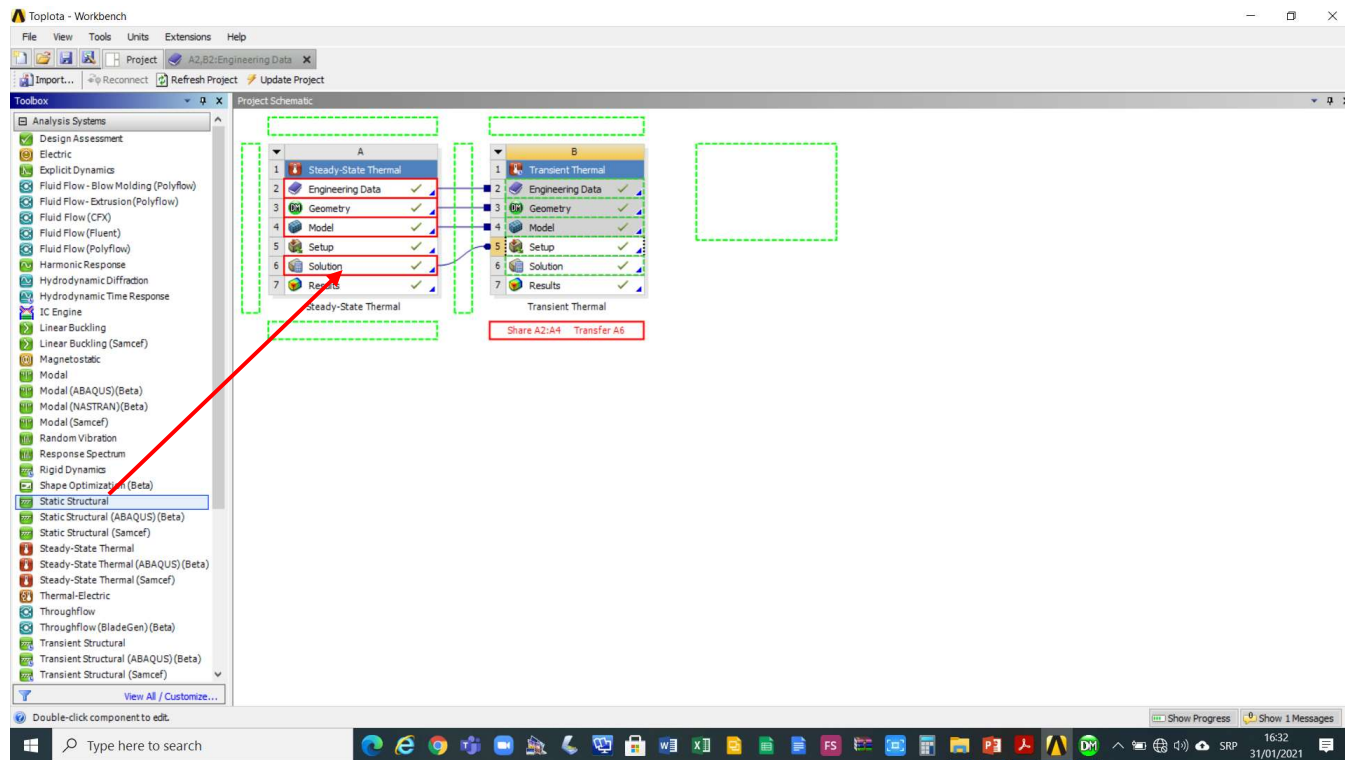
Toplotno opterećenje

Simulacija promjene raspodjele toplotnog fluksa u smjeru Y ose opcija *Graph*



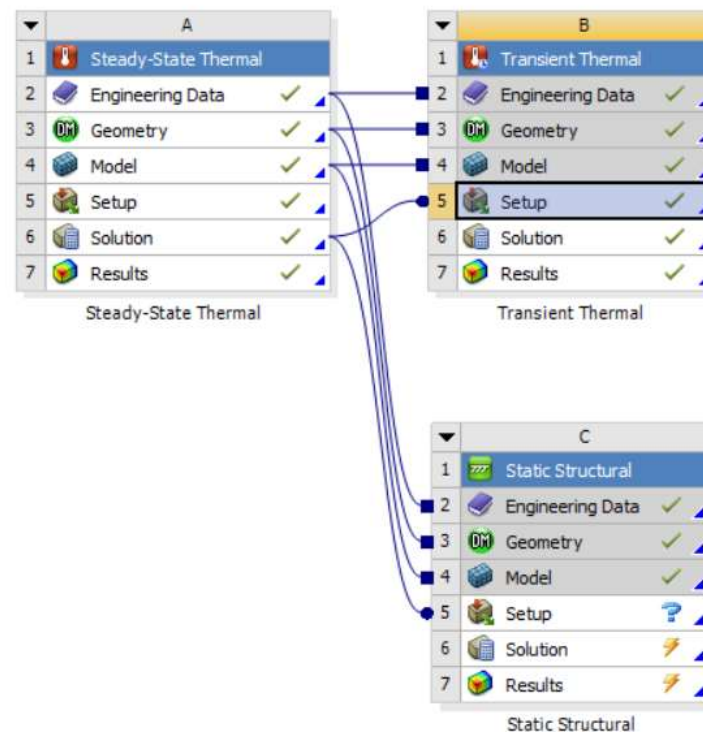
Toplotno opterećenje

Kreirati strukturnu analizu (*Static Structural*) povlačenjem na *Steady-State Thermal*->*Solution* stavku stacionarne analize



Toplotno opterećenje

Prethodni postupak omogućava potrebno povezivanje stacionarne termalne analize i strukturne analize



Toplotno opterećenje

Aktivirati modul Engineering Data i dodati podatke o gustini, modulu elastičnosti, Poisson-ovom koeficijentu, koeficijentu toplotnog širenja

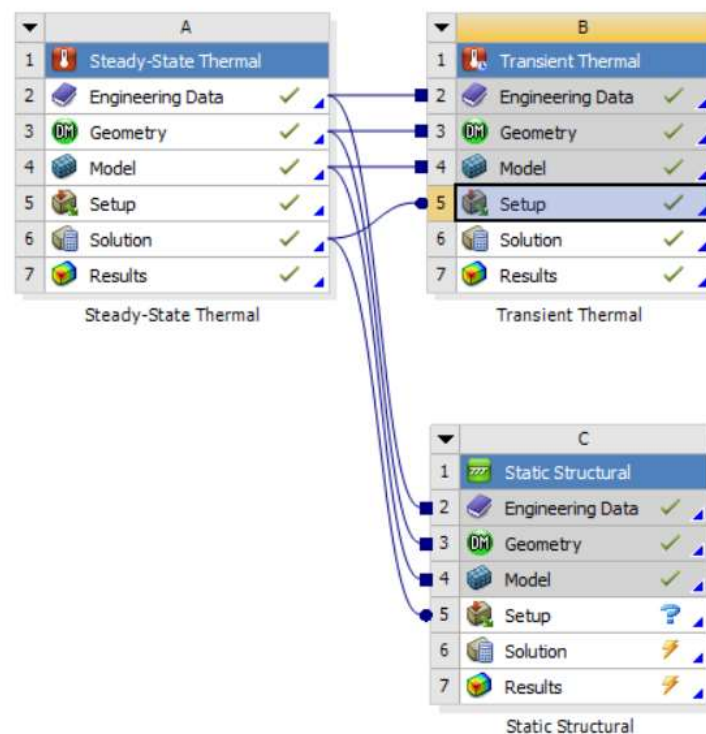
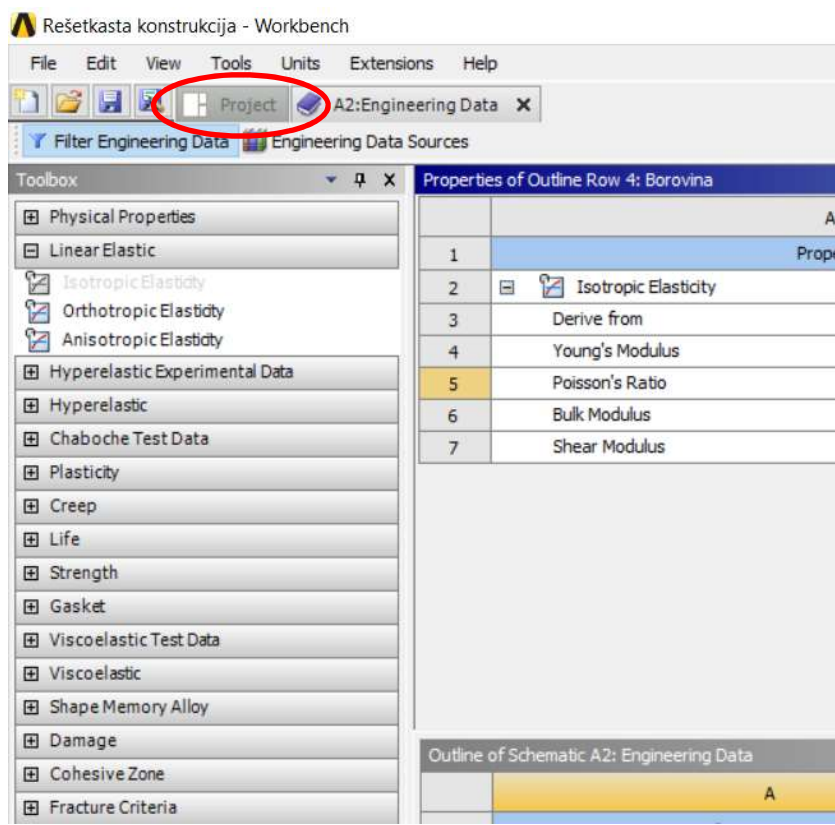
The screenshot displays the ANSYS Workbench Engineering Data environment. On the left, the 'Engineering Data' module is selected in the tree view. A context menu is open over the 'Engineering Data' node, showing options like 'Edit...', 'Duplicate', and 'Properties'. The main workspace is divided into several panes:

- Toolbox:** Lists various material models such as 'Physical Properties', 'Linear Elastic', and 'Viscoelastic'.
- Outline of Schematic:** Shows a table with columns A, B, C, and D. Row 3 is highlighted for 'Aluminum'.
- Properties of Outline Row 3: Aluminium:** A table listing material properties for Aluminum:

Property	Value	Unit
Density	2800	kg m ⁻³
Isotropic Instantaneous Coefficient of Thermal Expansion	2,2E-05	C ⁻¹
Isotropic Elasticity		
Derive from	Young's Modulus and ...	
Young's Modulus	7E+10	Pa
Poisson's Ratio	0,3	
Bulk Modulus	5,833E+10	Pa
Shear Modulus	2,692E+10	Pa
Isotropic Thermal Conductivity	170	W m ⁻¹ K ⁻¹
- Table of Properties:** A smaller table showing the 'Coefficient of Thermal Expansion (C⁻¹)' as 2,2E-05.
- Chart of Properties:** A graph titled 'Coefficient of Thermal Expansion' showing a constant value of approximately 0.000022 across a temperature range from -1 to 1 °C.

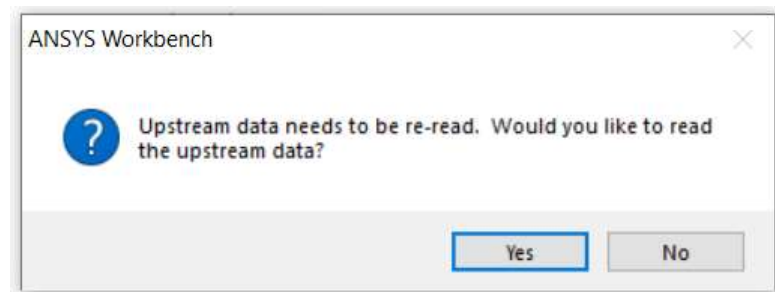
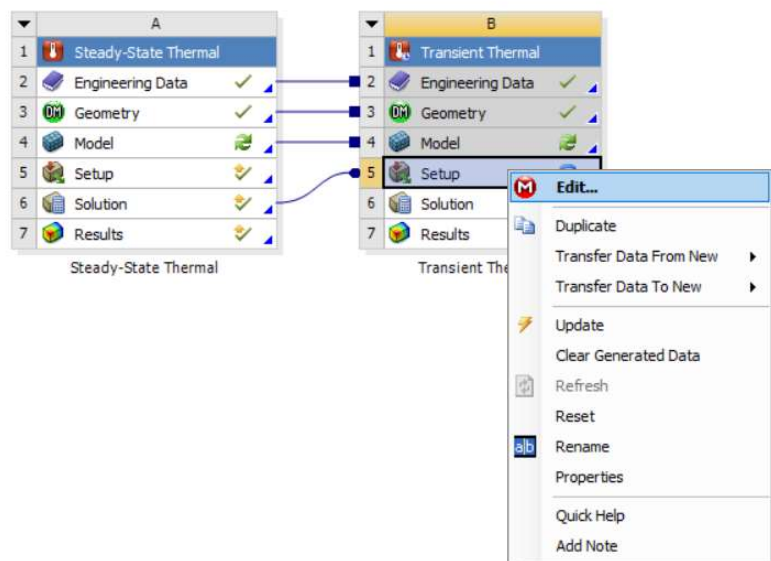
Toplotno opterećenje

Izabrati opciju *Project* za povratak na shemu projekta



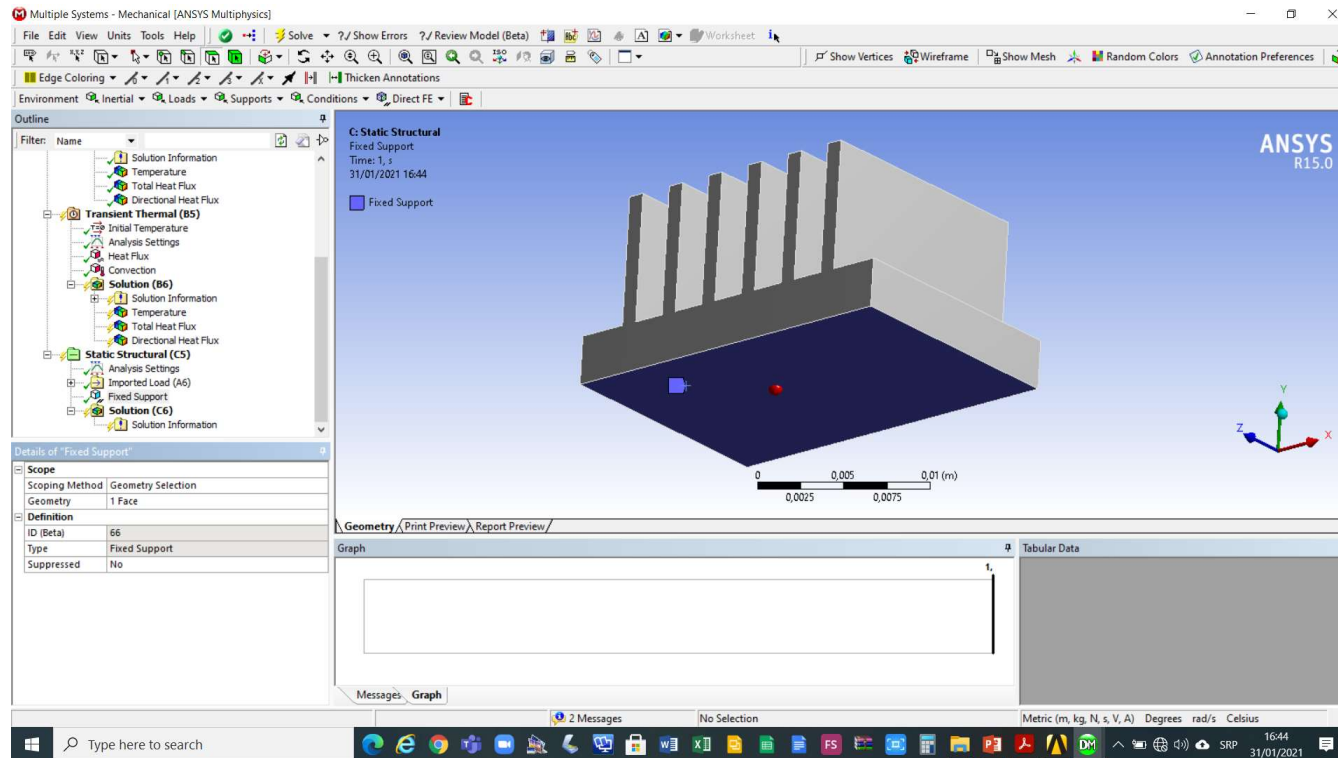
Toplotno opterećenje

Aktivirati modul Multiple Systems (*Static Structural*->*Setup*->*Edit*) i odgovoriti sa *Yes* kako bi se učitali novounešeni podaci



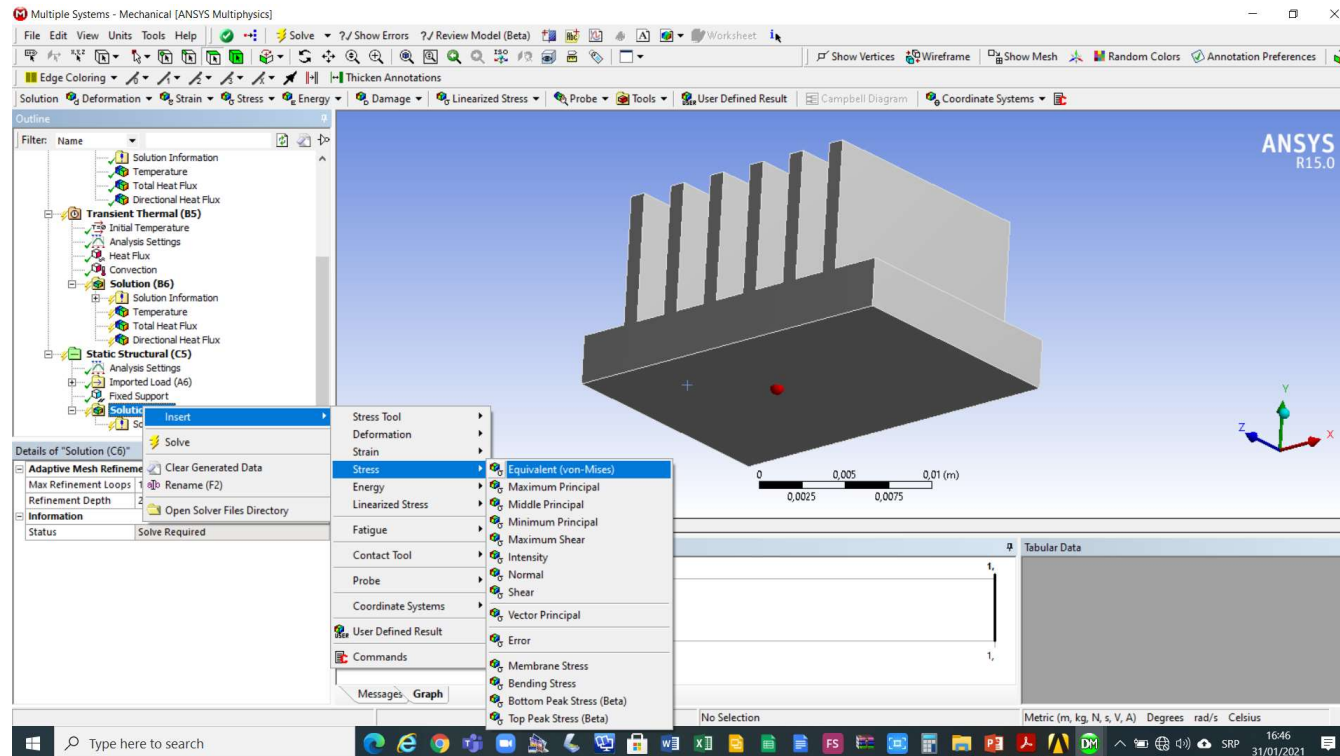
Toplotno opterećenje

Dodati nepokretni oslonac na donju površinu
Static Structural->Insert->Fixed Support



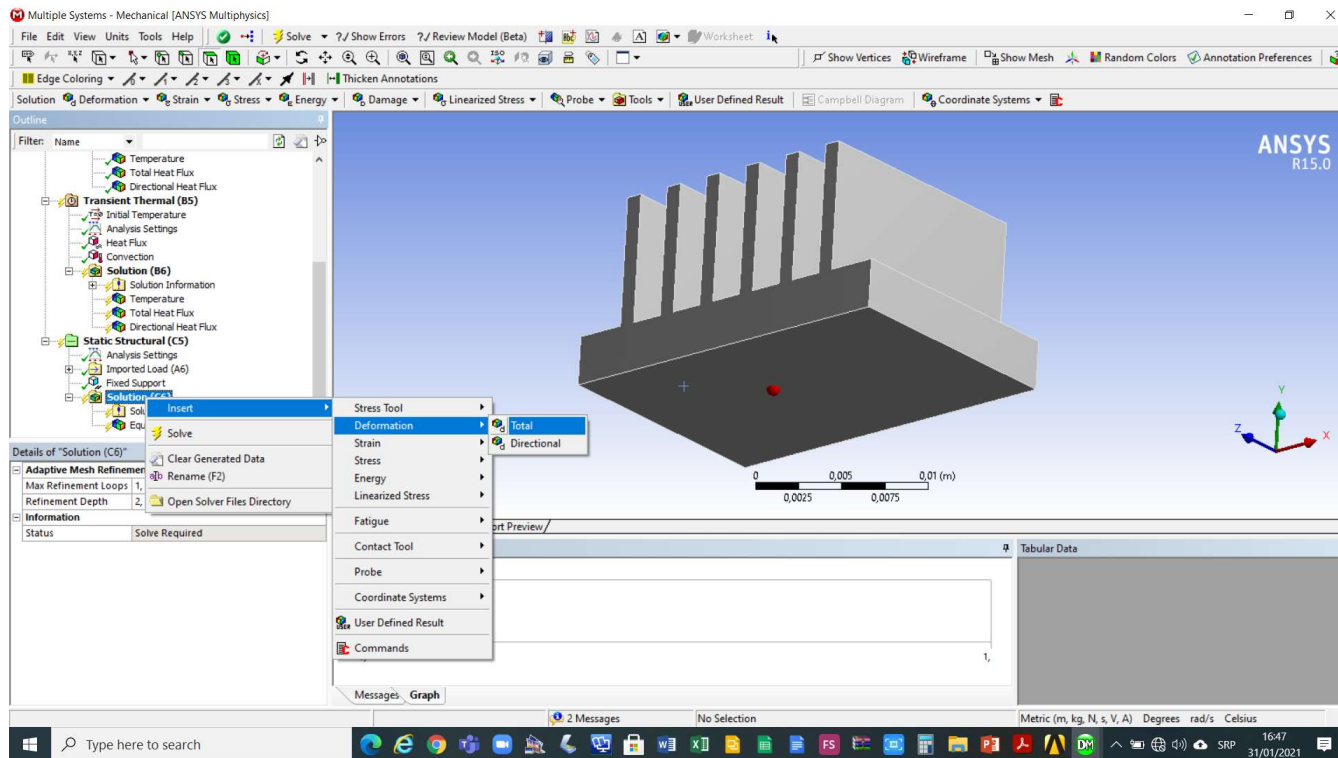
Toplotno opterećenje

Izabrati analizu koja se želi realizovati *Solution->Insert->Stress->Equivalent von Mises*



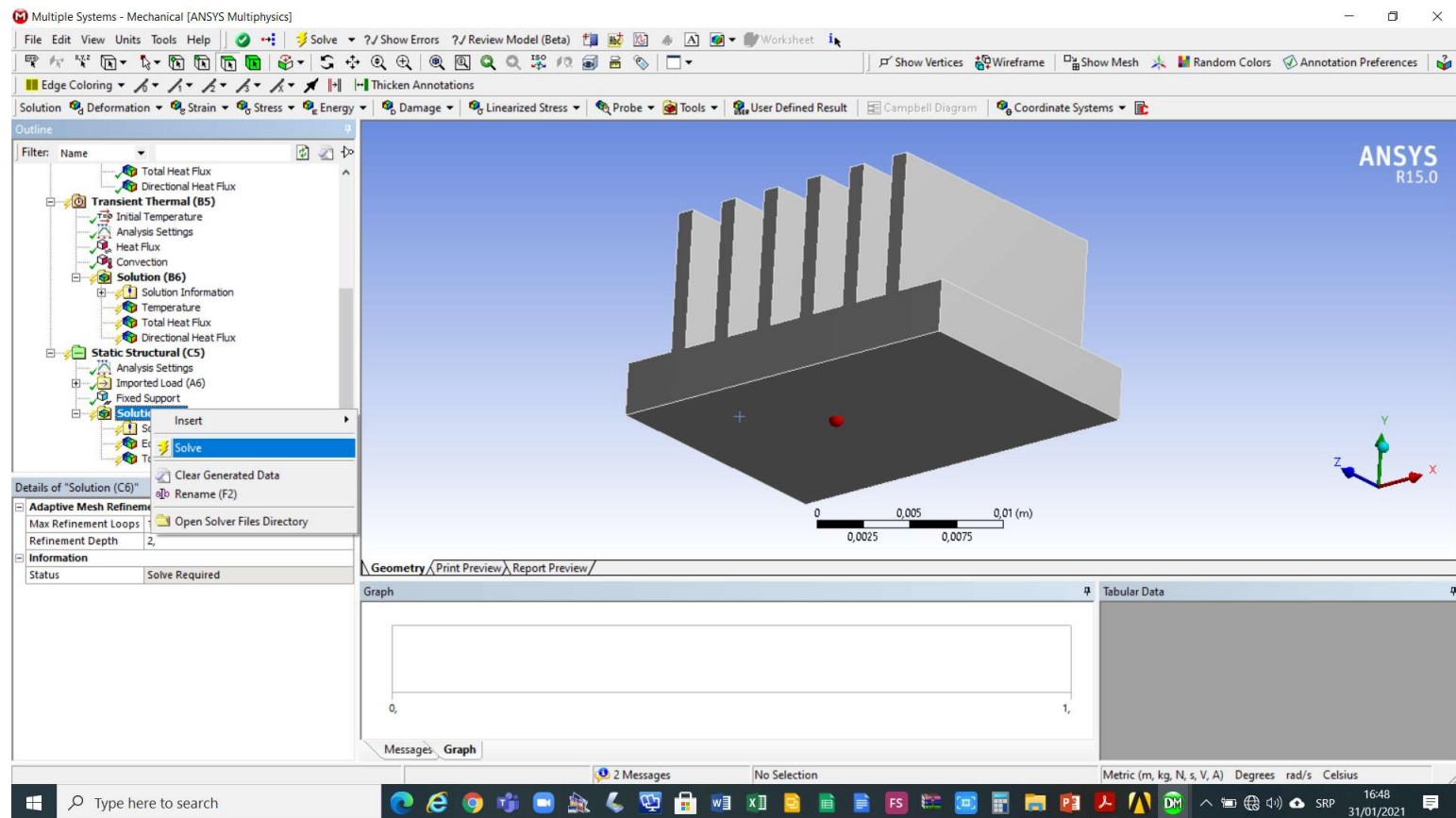
Toplotno opterećenje

Izabrati analizu koja se želi realizovati *Solution->Insert->Stress->Total Deformation*



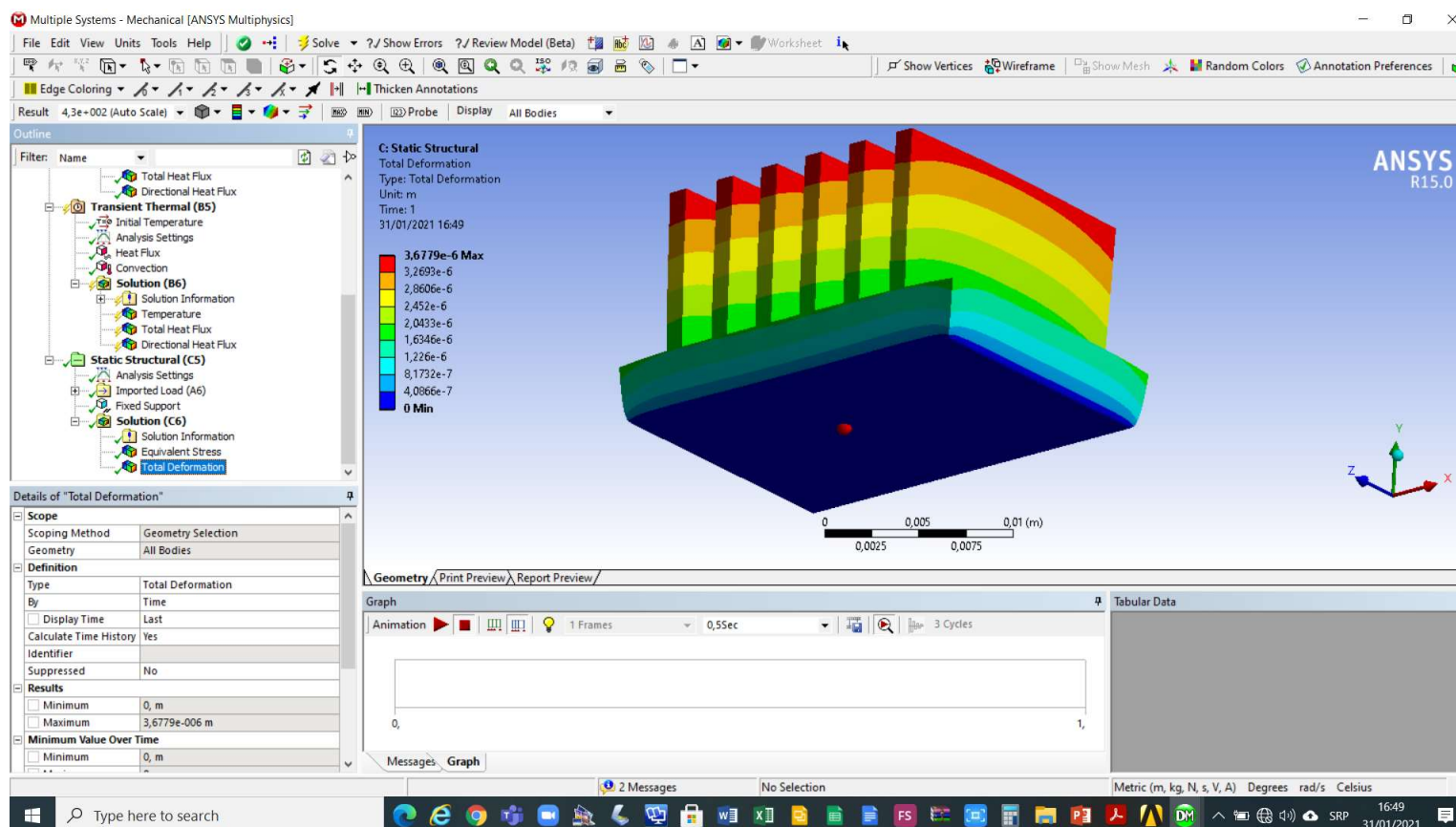
Toplotno opterećenje

Aktivirati izvršenje analize *Solution*->*Solve*



Toplotno opterećenje

Raspodjela toplotne deformacije



Toplotno opterećenje

Raspodjela toplotnih von Mises-ovih napona

