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 W/(m·K), specifična toplota c=870 J/(kg·K), E=70 GPa, ν=0.3, ρ=2800 kg/m^3, koeficijent toplotnog širenja α=22·10^{-6} 1/^{0}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 ^{0}C sa koeficijentom prenosa toplote h=30 W/(m^2· ^{0}C), . Odrediti:

- Raspodjelu temperature pri stacionarnom stanju ukoliko je početna temperature elementa za odvod toplote T=28 ^{0}C, konstatni ulazni toplotni fluks po donjoj površini q=1000 W/m^2

Postavka zadatka

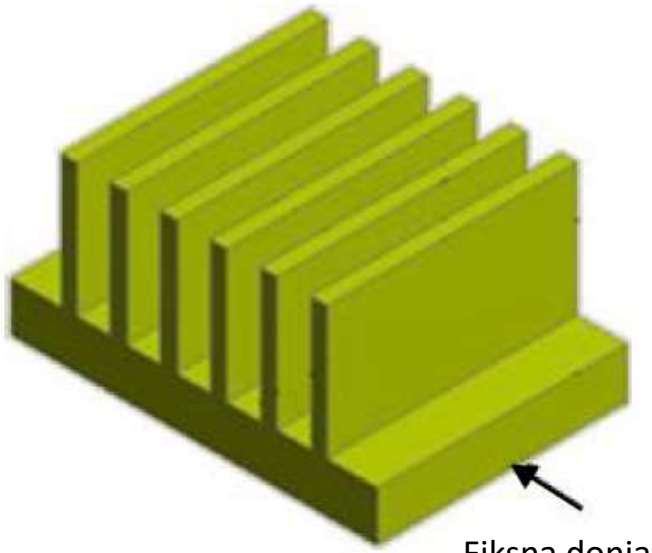
- Raspodjelu temperature pri nestacionarnom stanju ukoliko je ulazni toplotni fluks kvadratna funkcija sa periodom τ=90 s i minimalnom, odnosno, maksimalnom veličinom q=(0÷ 1000) W/m^2. Početno stanje raspodjele temperature odgovara raspodjeli dobijenoj u prethodnom koraku. Promjenu rapodjele temerature pratiti t=180 s.

- Raspodjelu toplotnih napona u stacionarnom stanju.

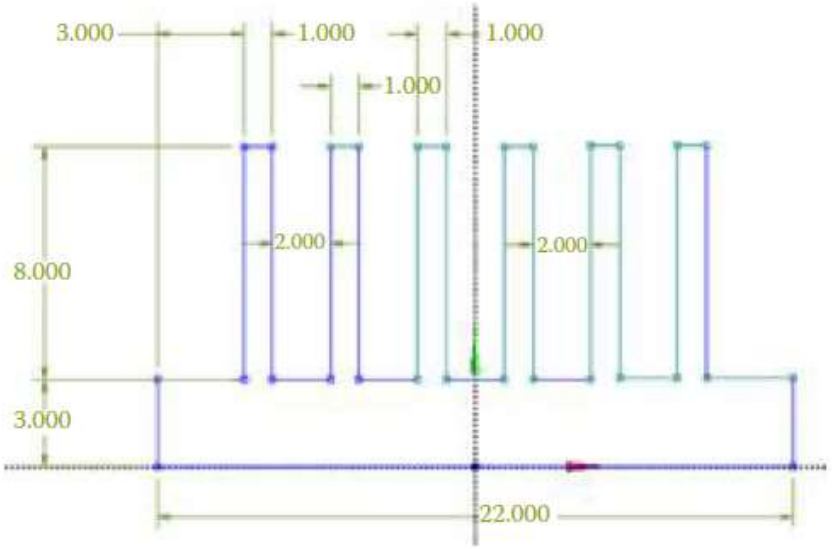
Postavka zadatka

Granični uslovi: Nepokretan oslonac po donjoj površini

Opterećenje: Toplotni fluks po donjoj površini



Fiksna donja površina

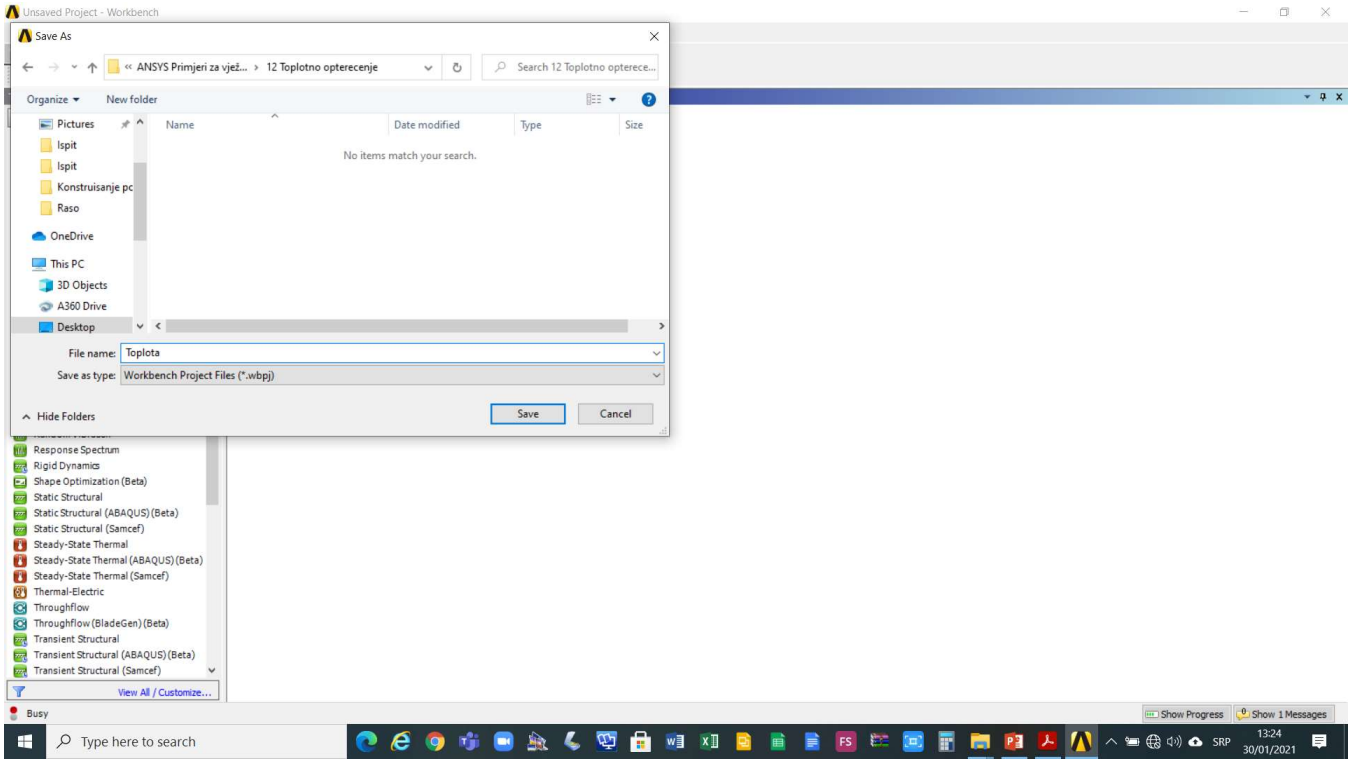


Sve dimenzije su date u mm

Postavka zadatka

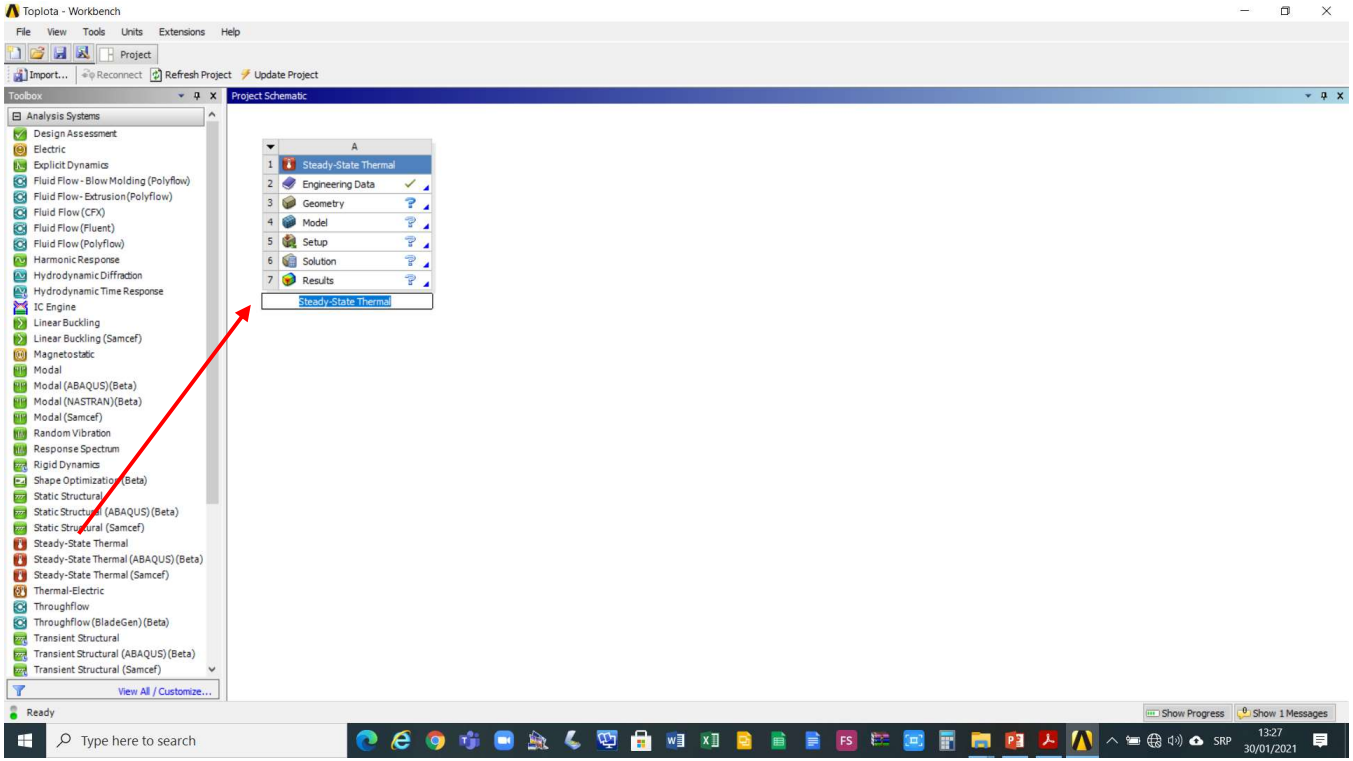
Toplotno opterećenje

Aktivirati program ANSYS i sačuvati prazan projekat pod nazivom Toplota



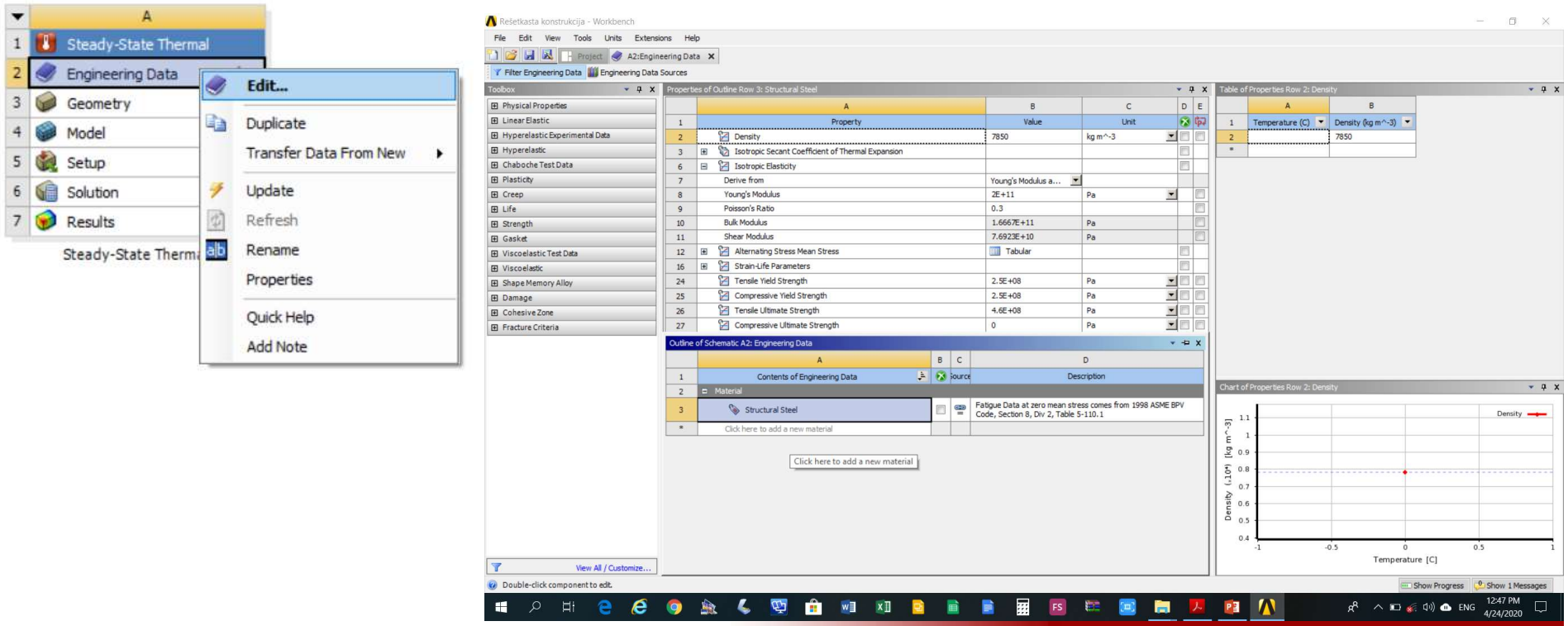
Toplotno opterećenje

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



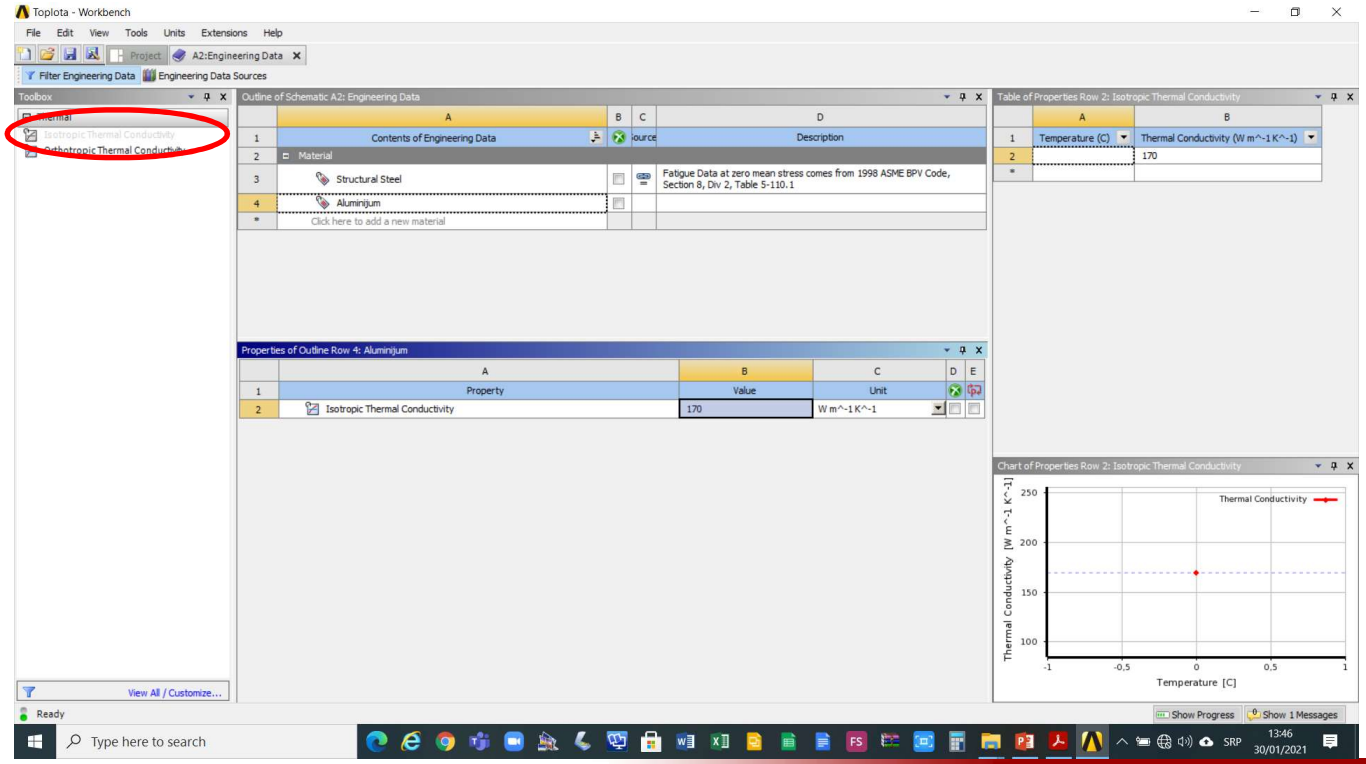
Toplotno opterećenje

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



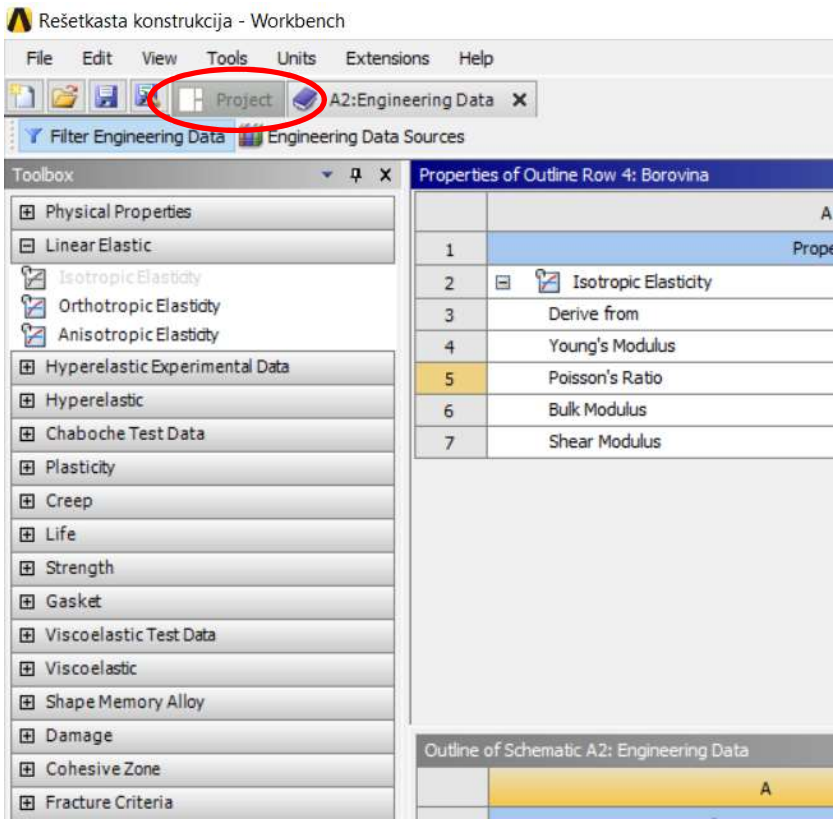
Toplotno opterećenje

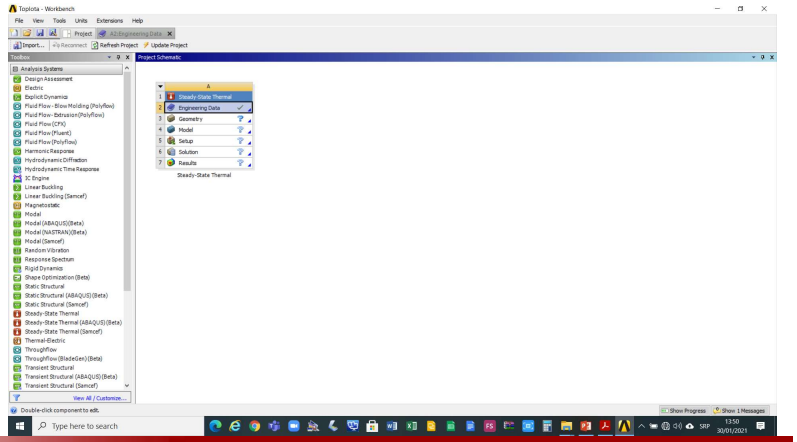
Promjeniti jedinice i unijeti karakteristiku novog izotropnog materijala (*Isotropic Thermal Conductivity*)



Toplotno opterećenje

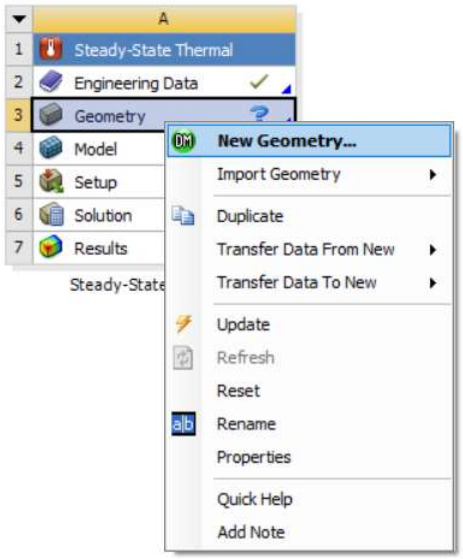
Izabrati opciju *Project* za povratak na shemu projekta

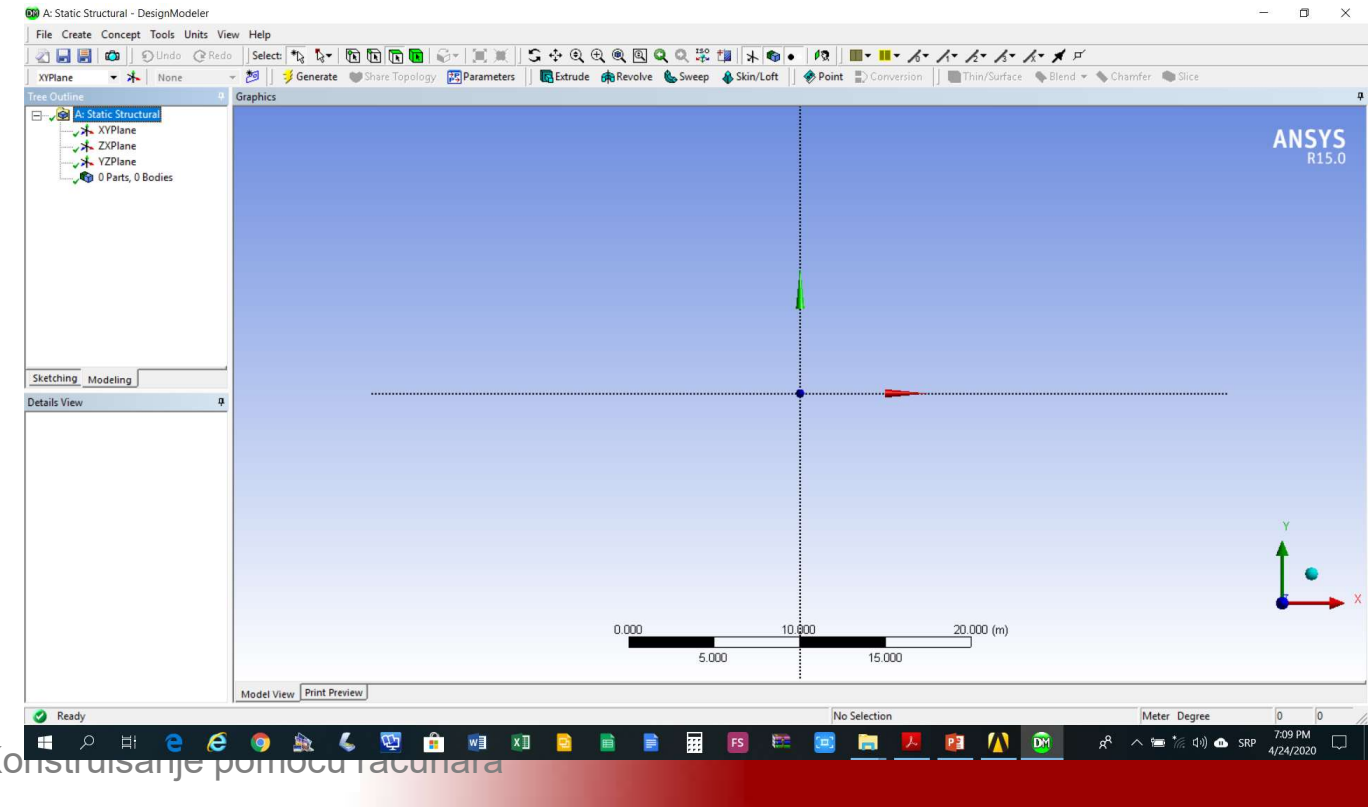




Toplotno opterećenje

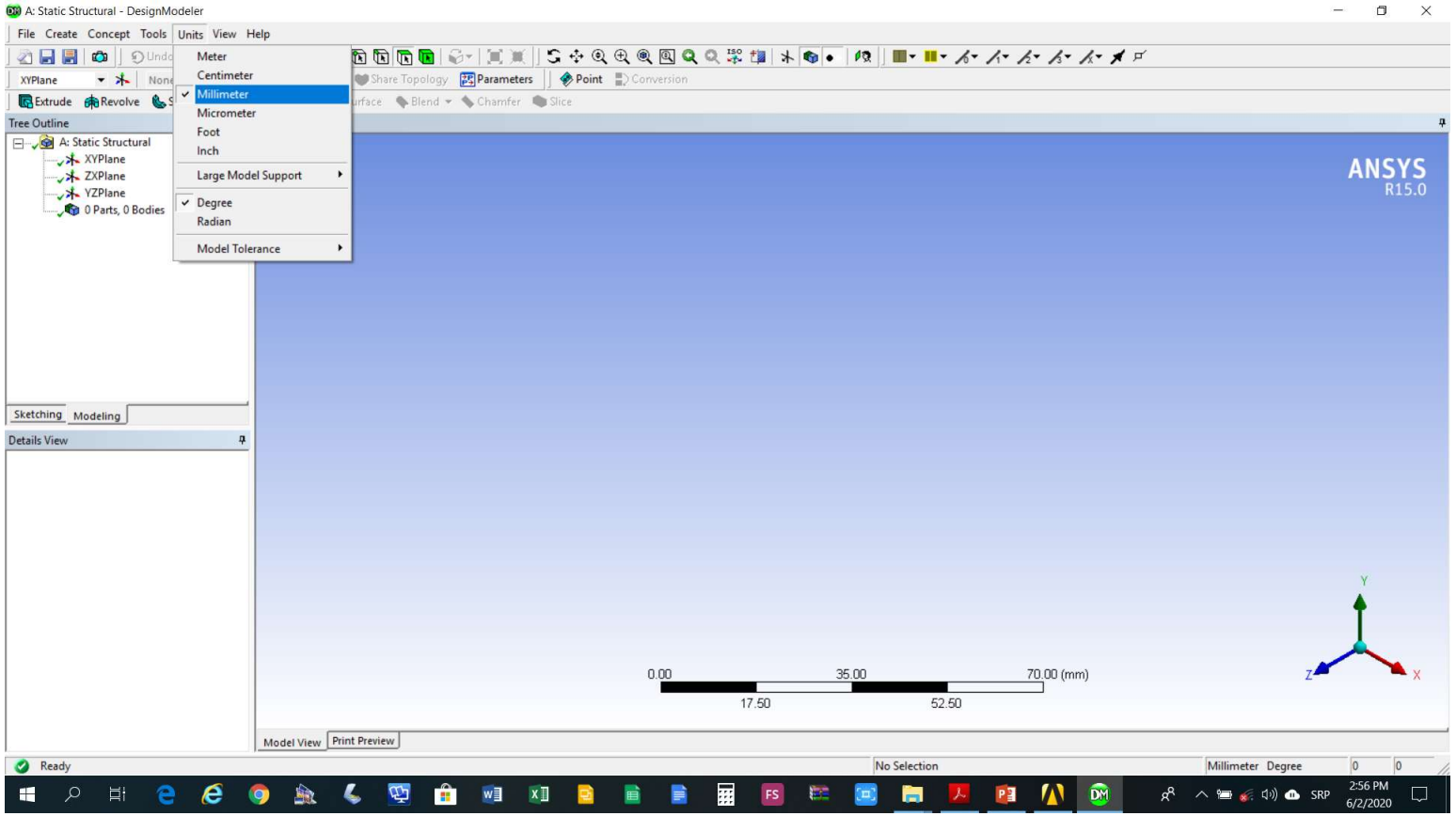
Aktivirati modul Design Modeler (*Geometry->New Geometry*)





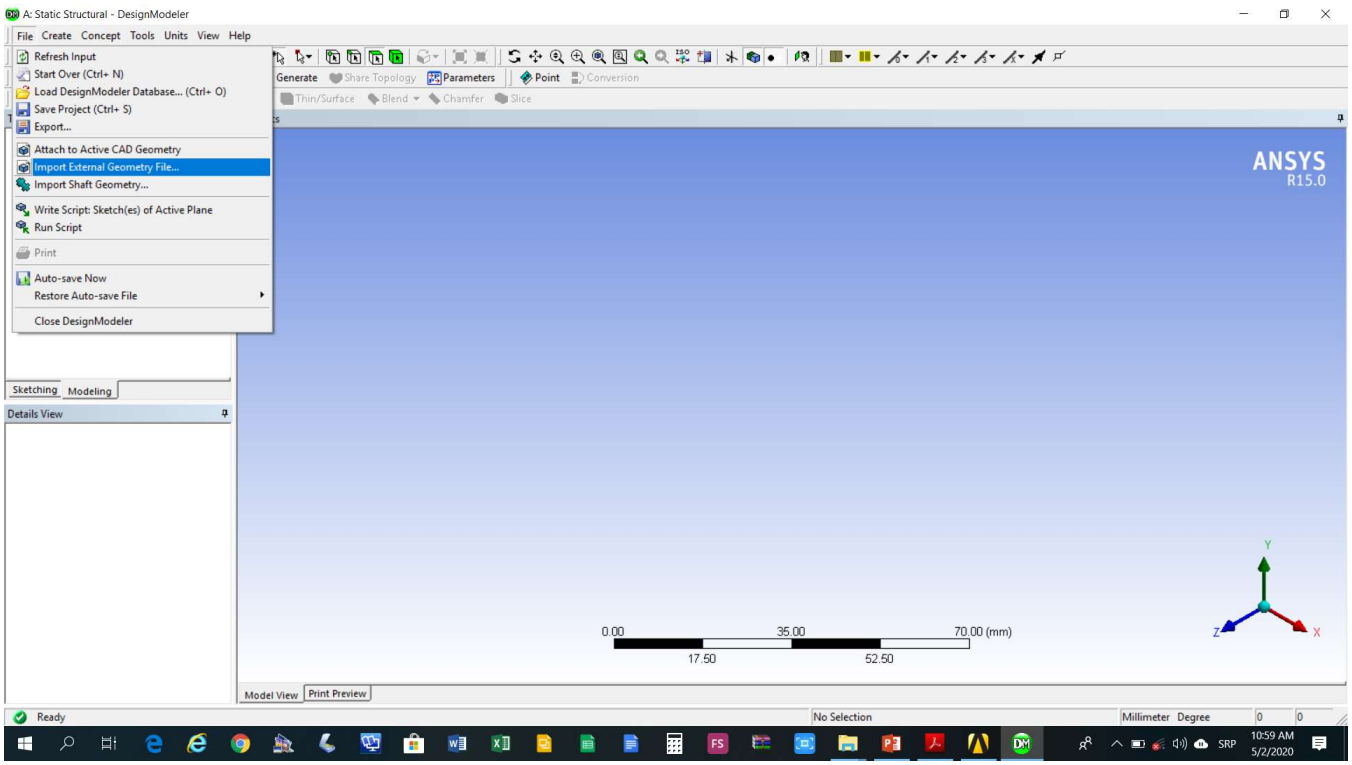
Toplotno opterećenje

Podesiti dužinske jedinice (Units->*Milimeter*)



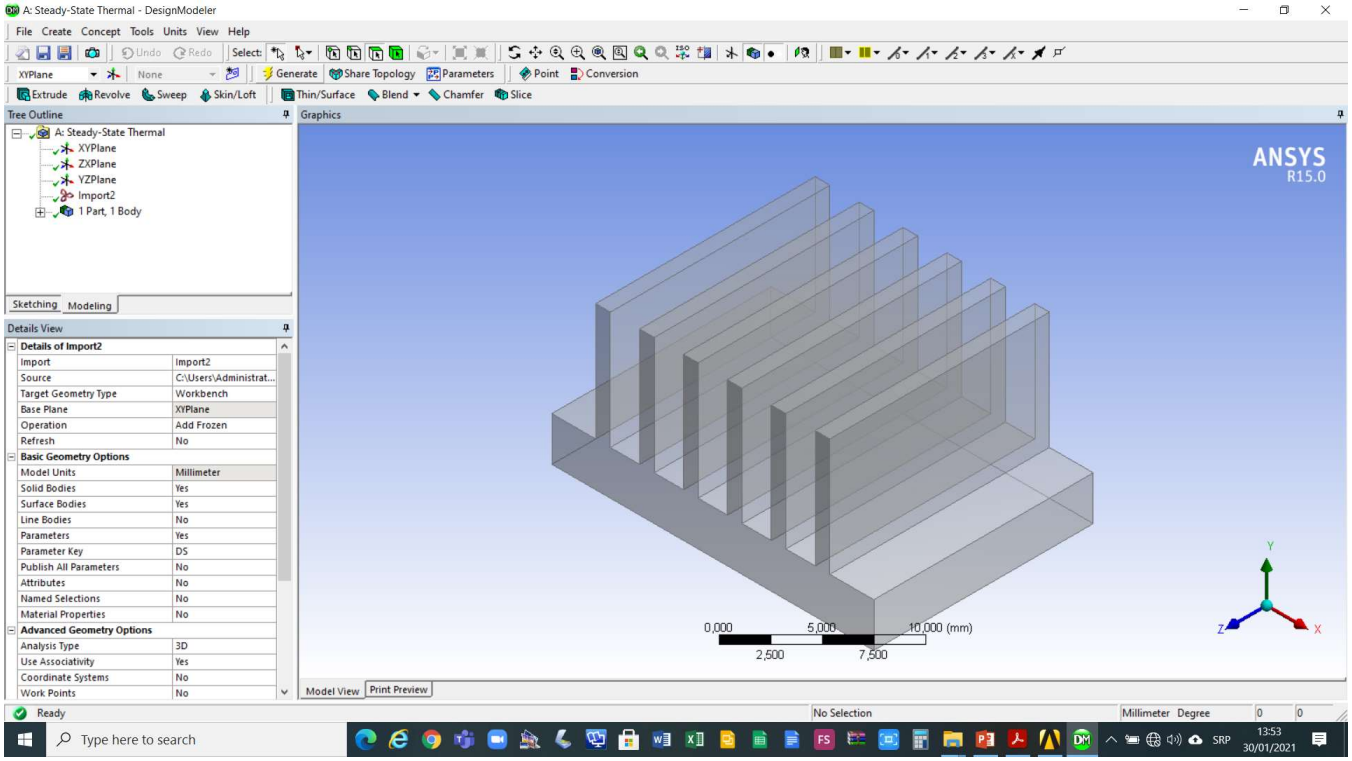
Toplotno opterećenje

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



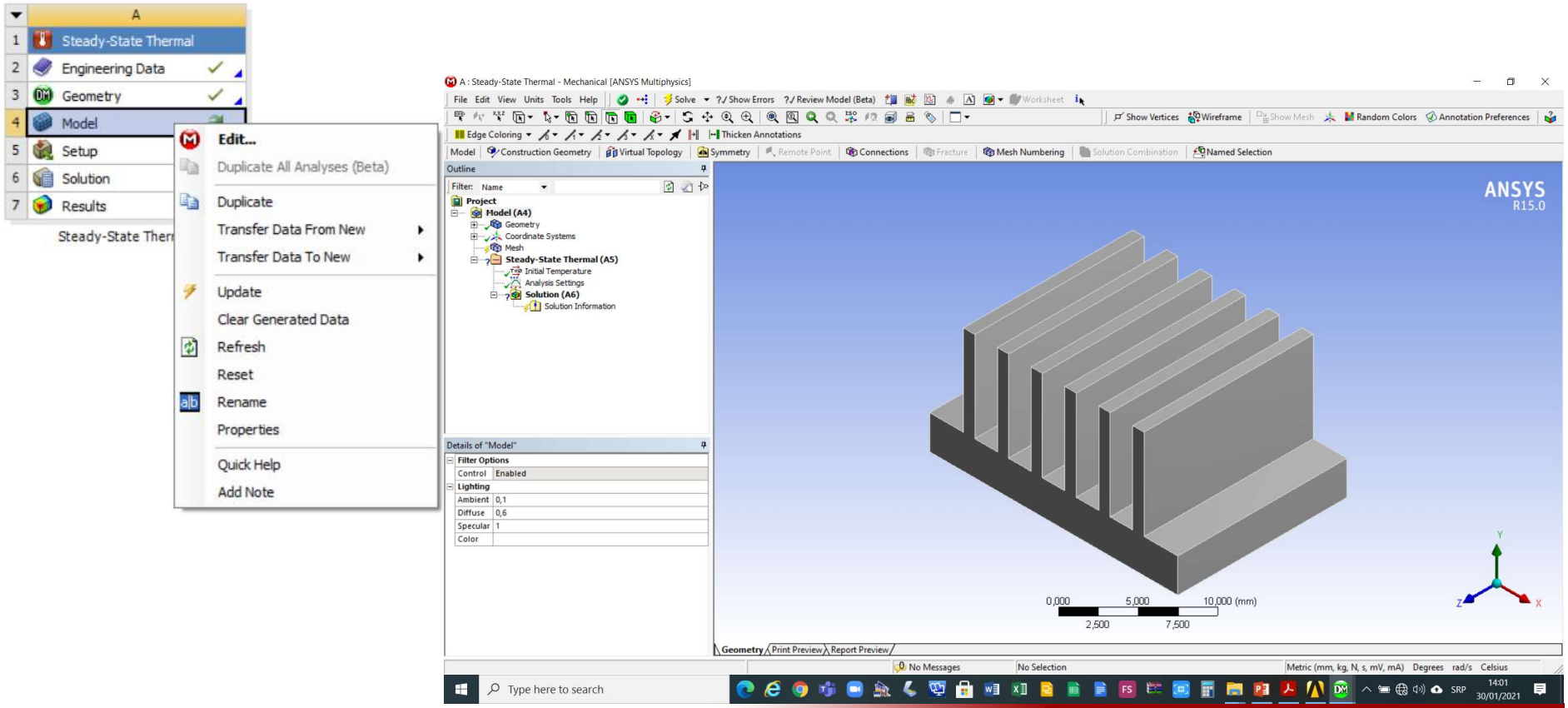
Toplotno opterećenje

Učitavanje eksterno generisane geometrije okončati komandom *Generate*



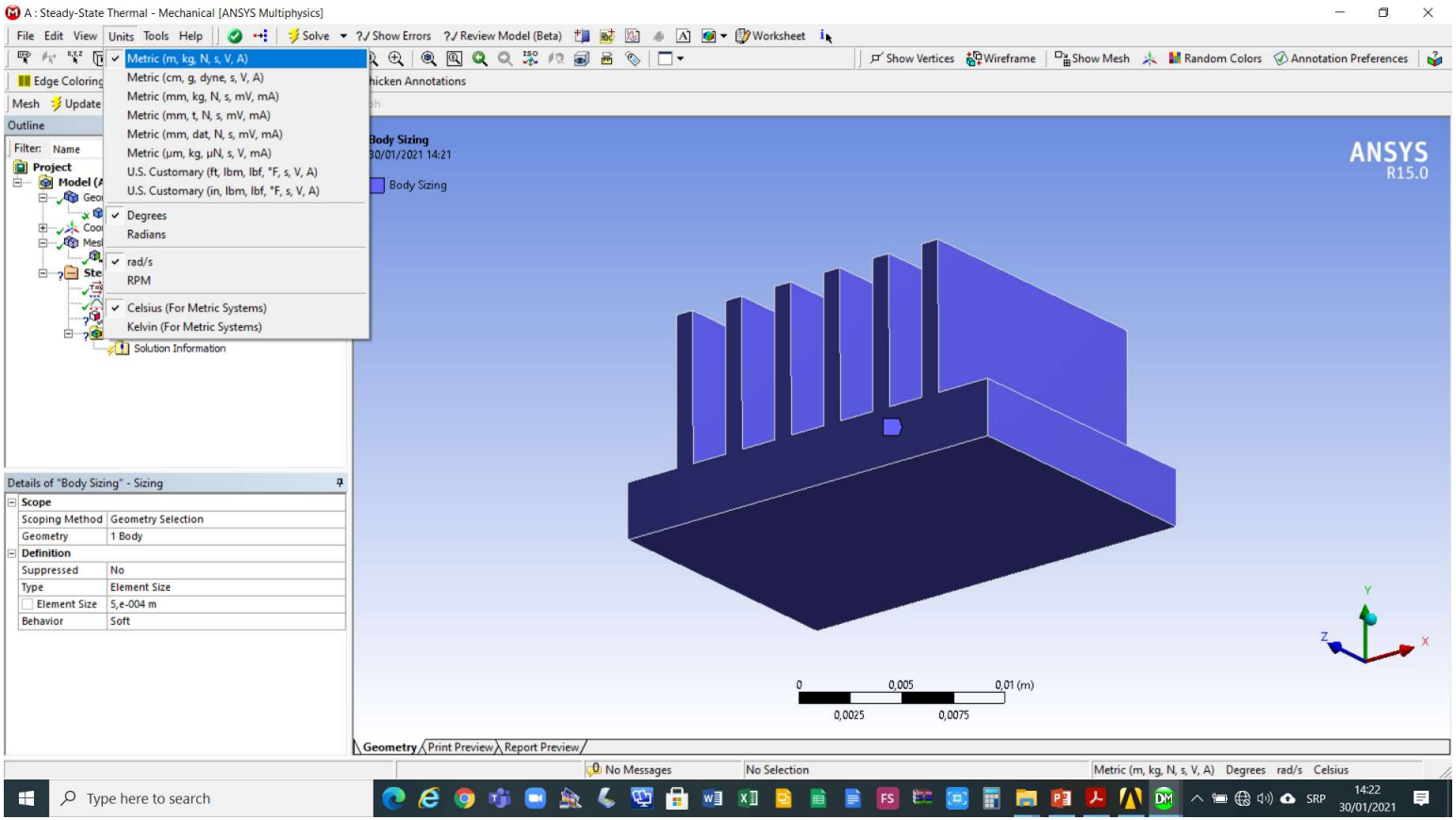
Toplotno opterećenje

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



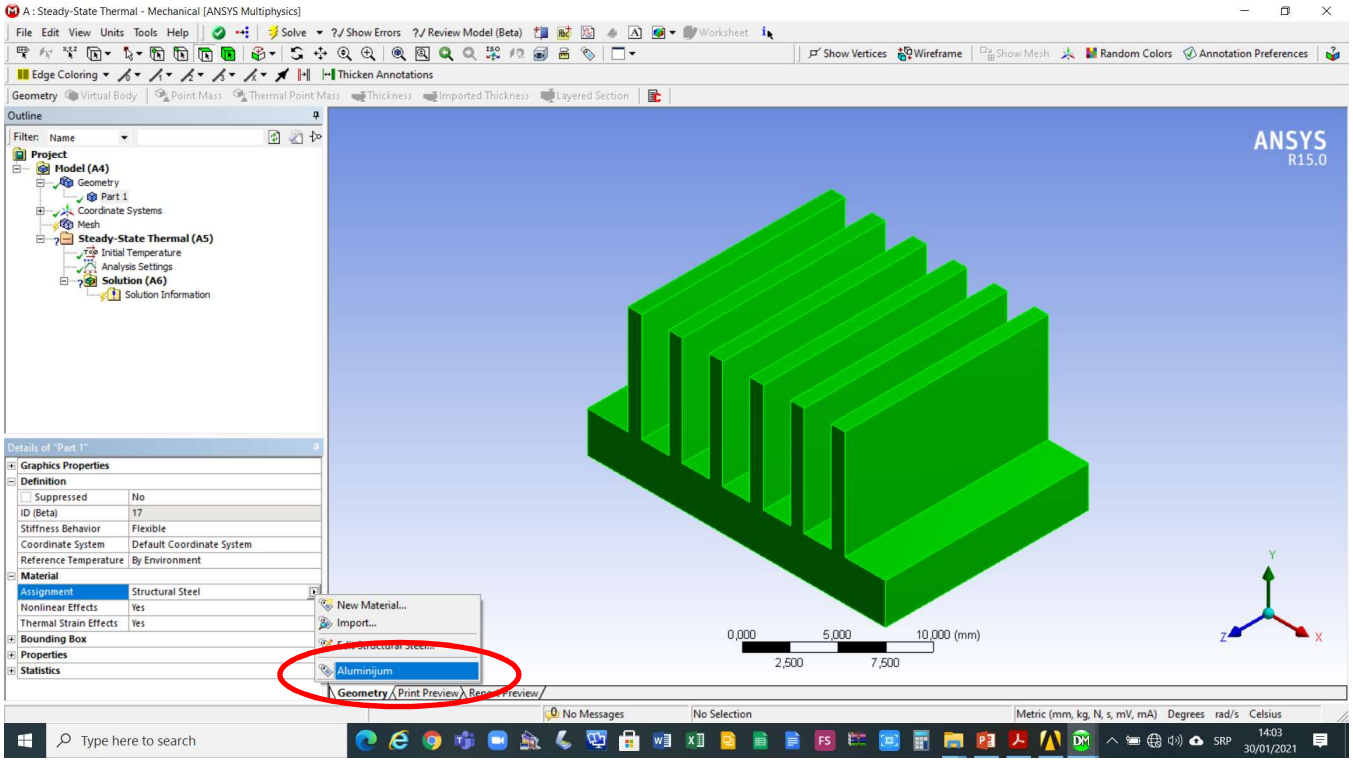
Toplotno opterećenje

Podesiti dužinske jedinice (*Units->Meter*)



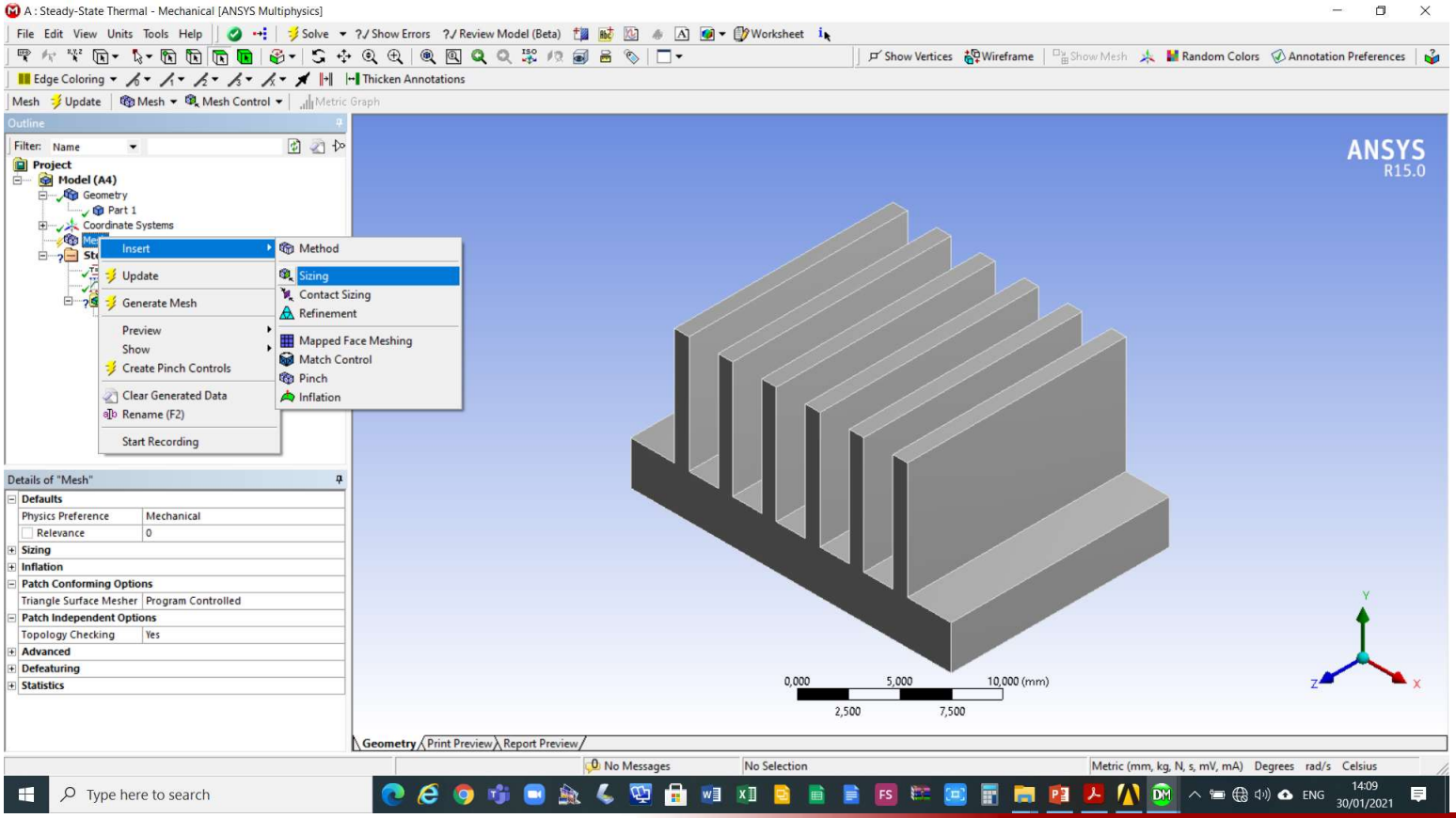
Toplotno opterećenje

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



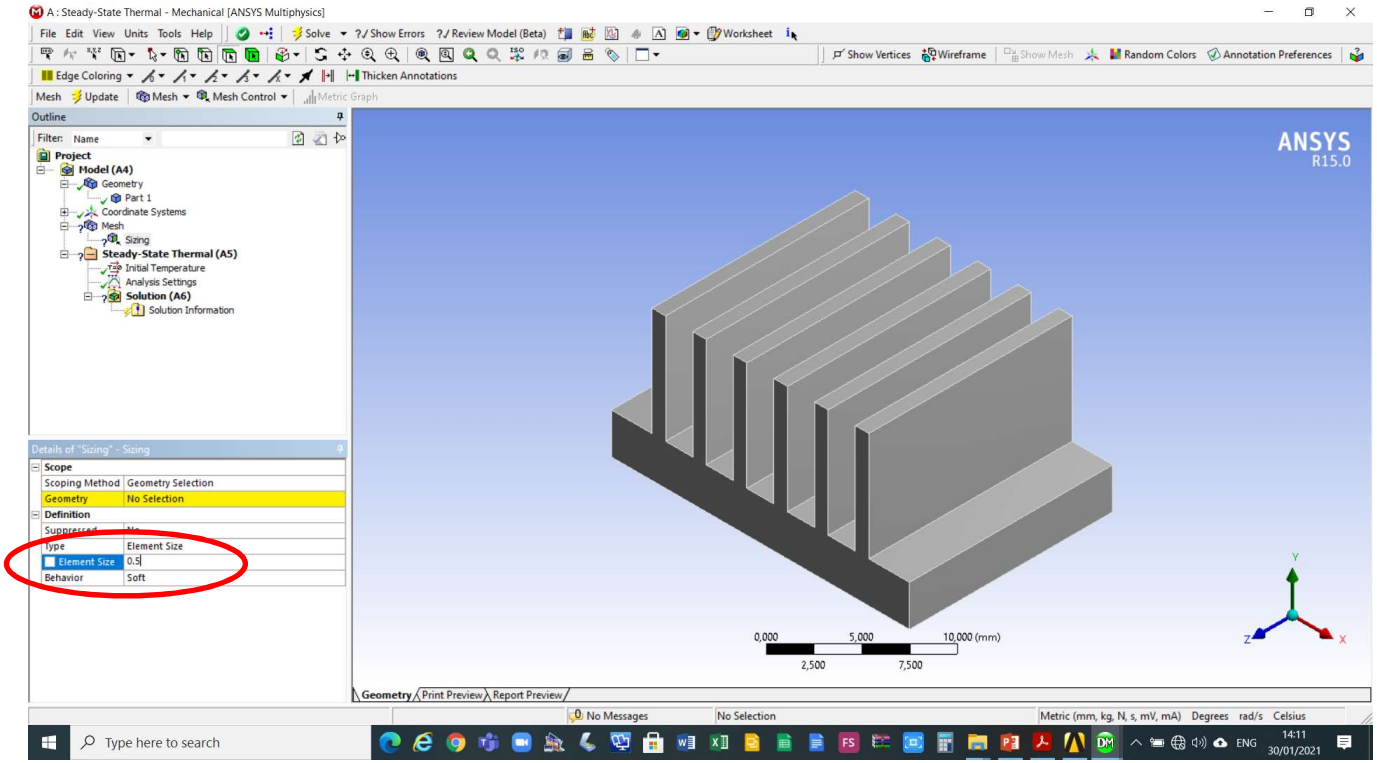
Toplotno opterećenje

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



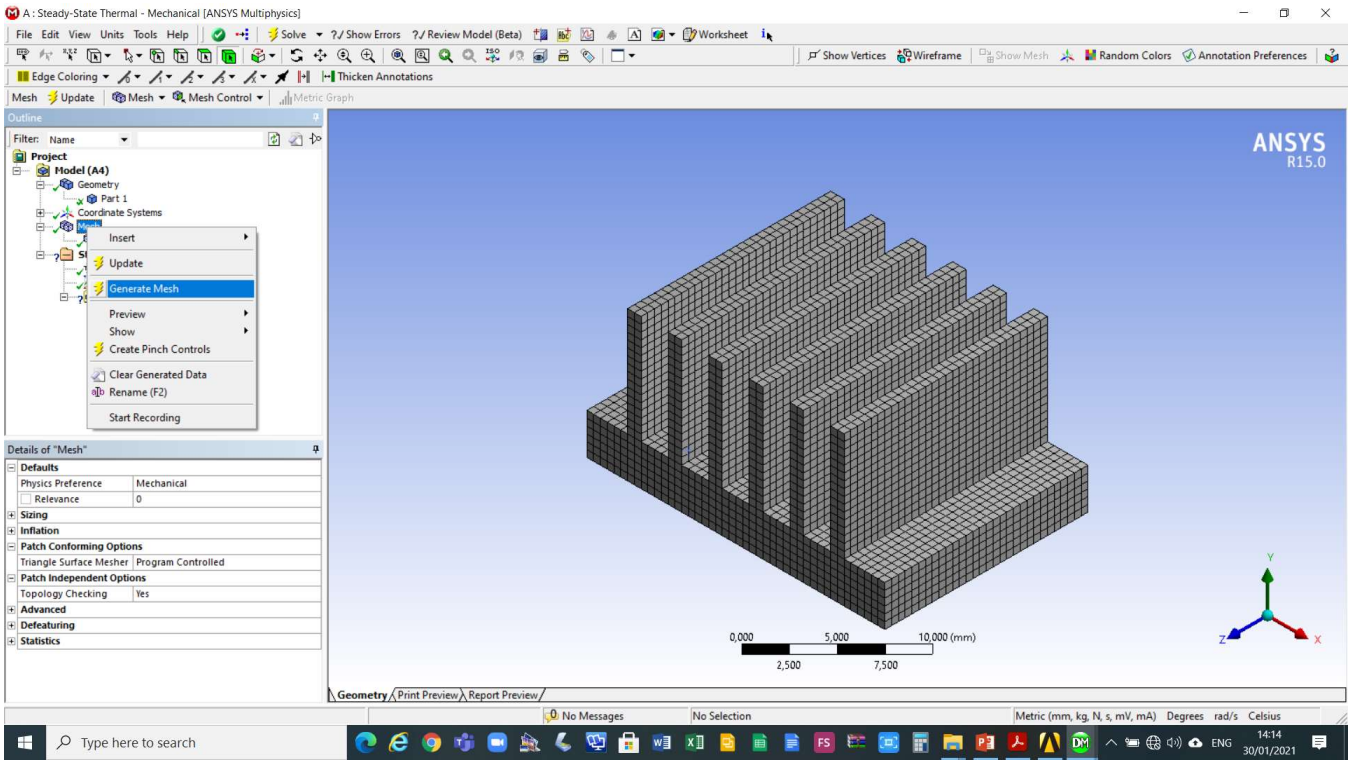
Toplotno opterećenje

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



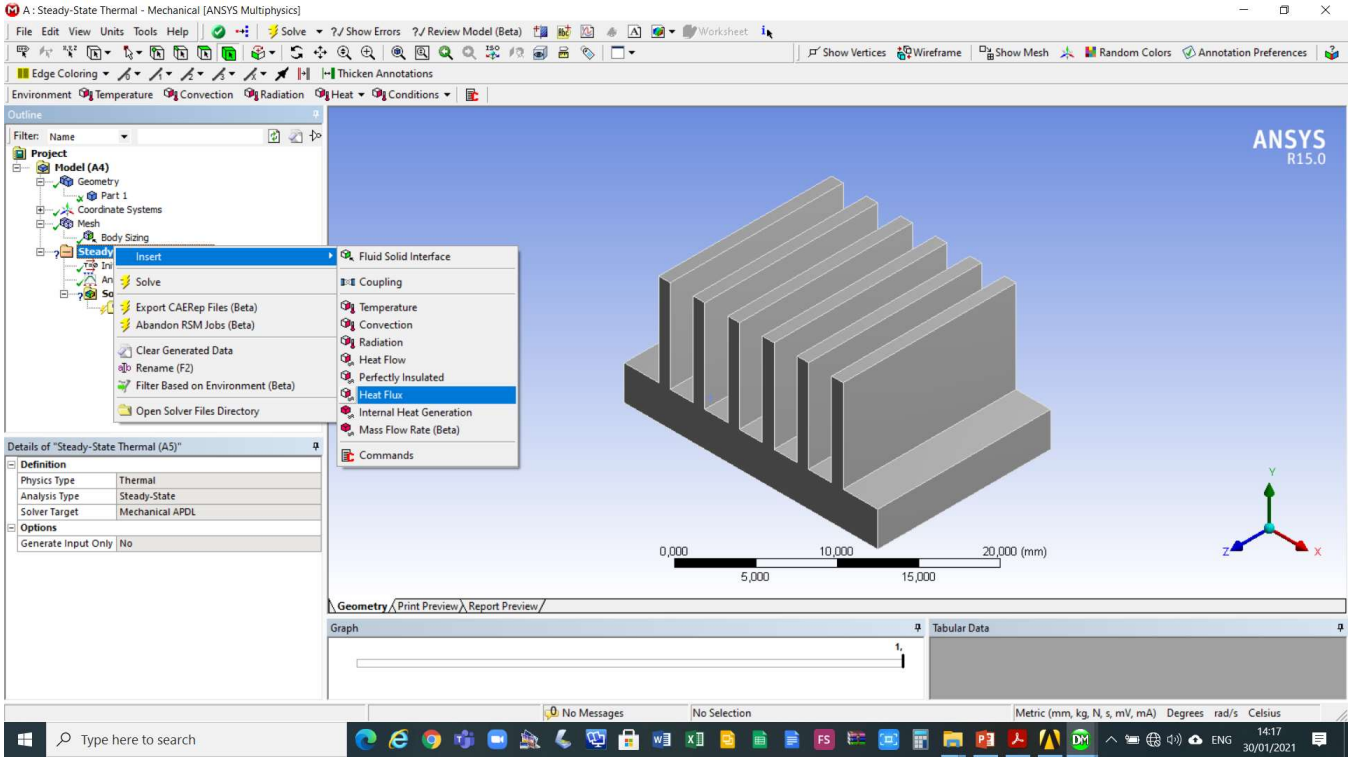
Toplotno opterećenje

Generisati mrežu konačnih elemenata *Mesh->Generate Mesh*



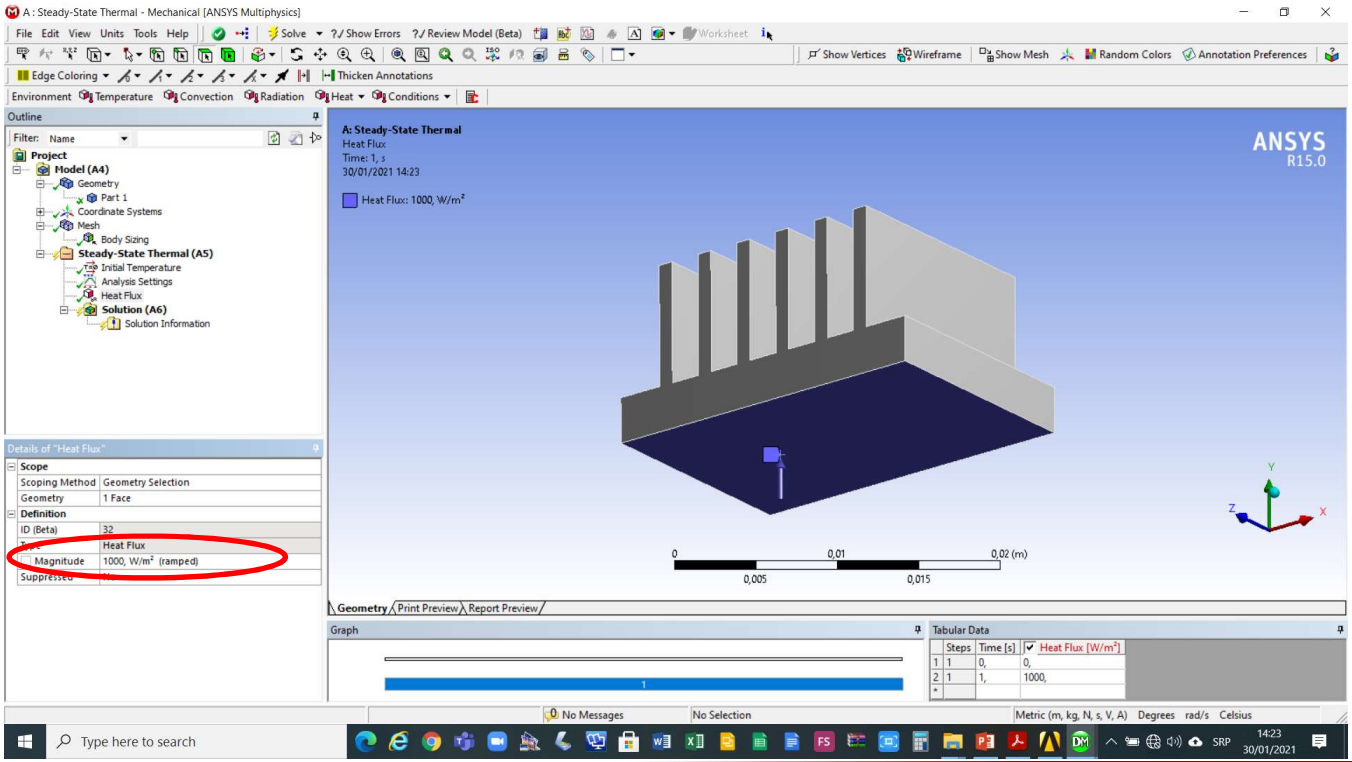
Toplotno opterećenje

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



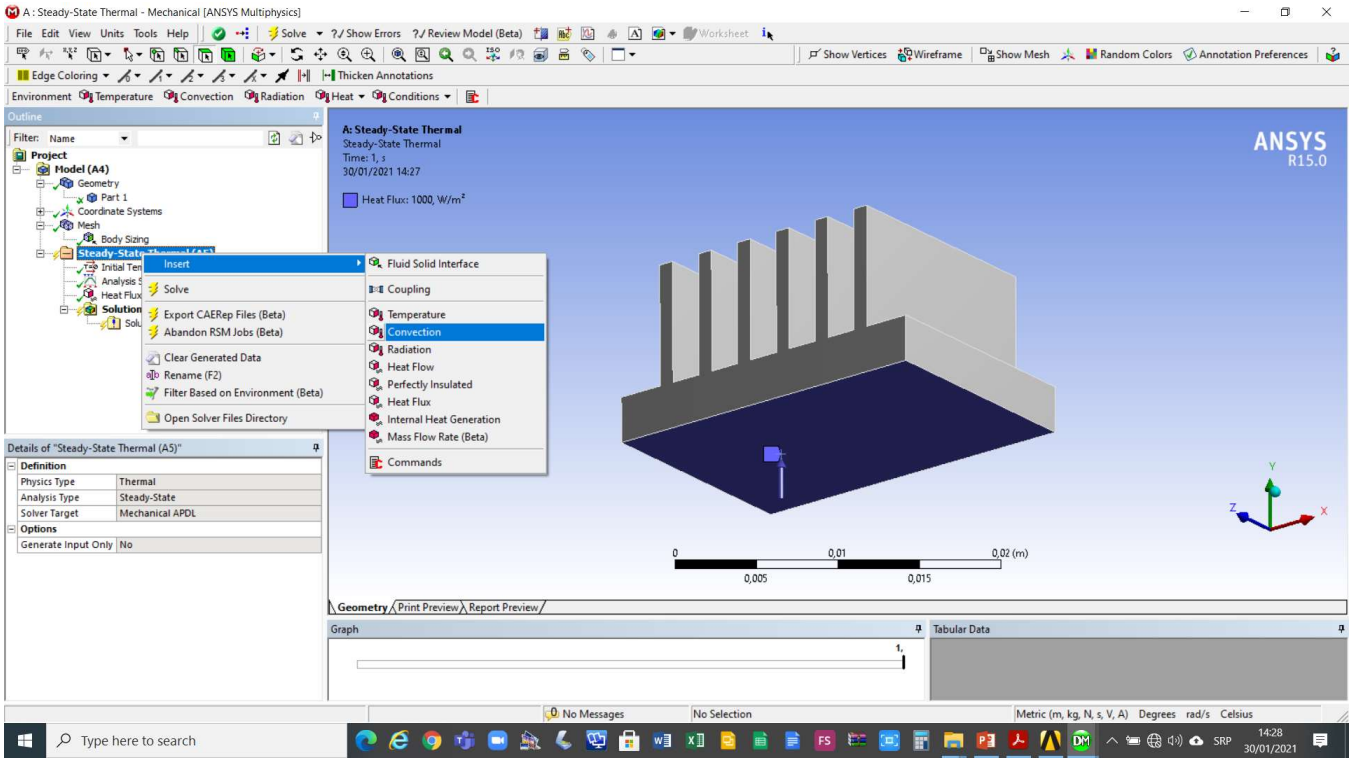
Toplotno opterećenje

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



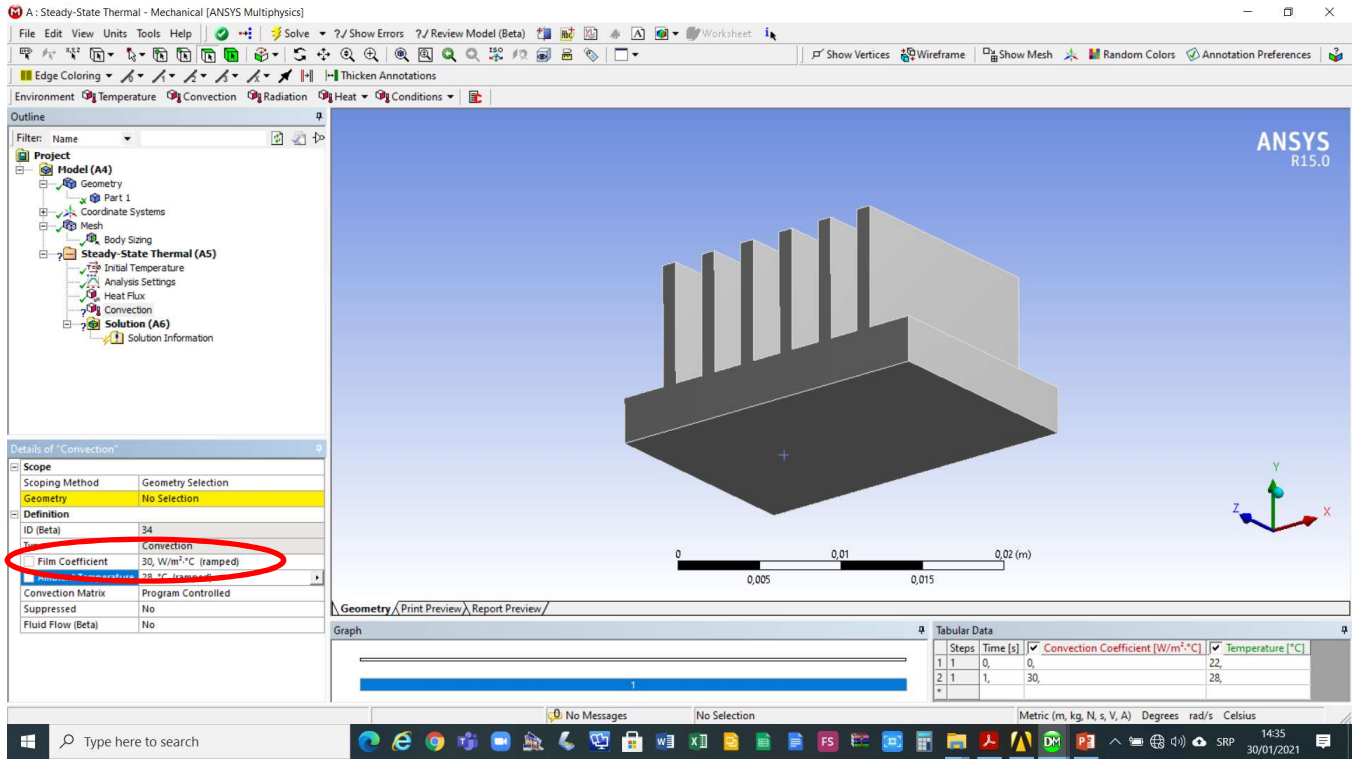
Toplotno opterećenje

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



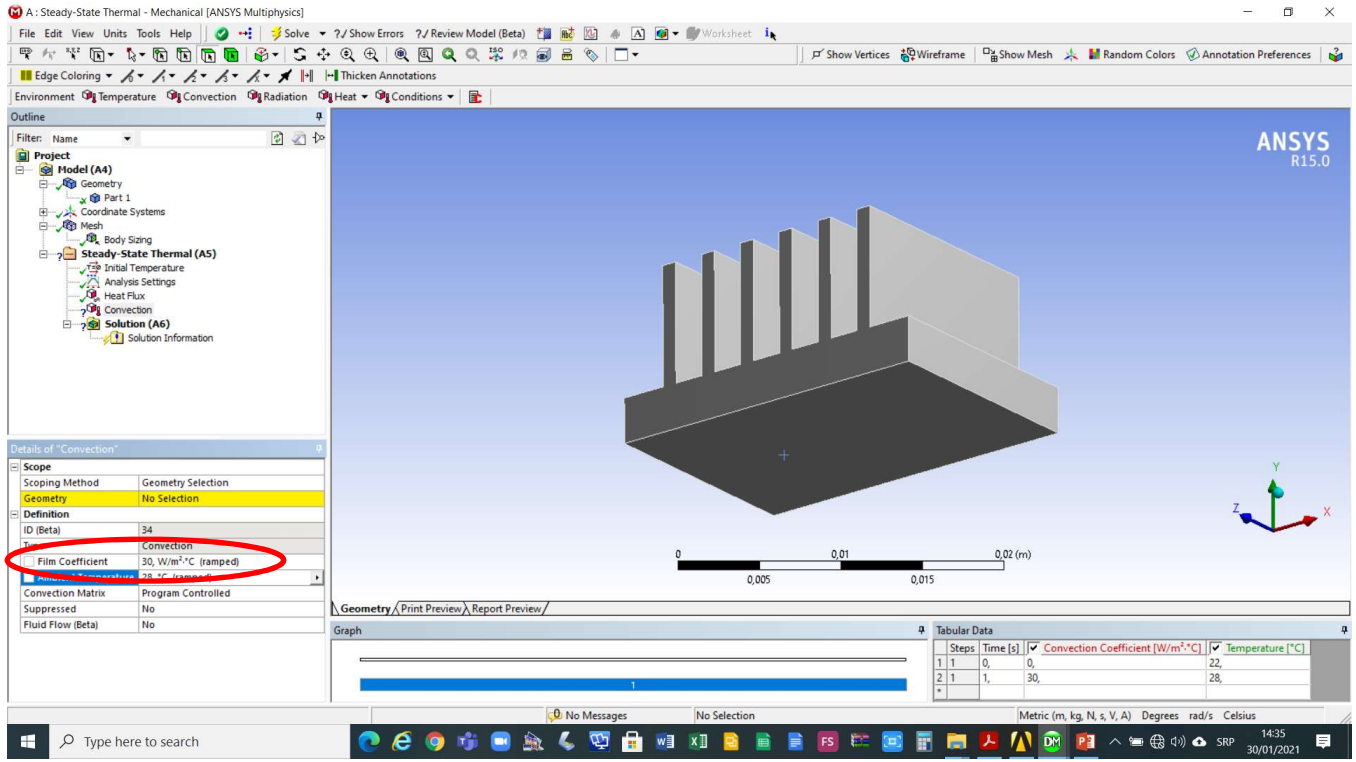
Toplotno opterećenje

Podešavanje koeficijenta prenosa toplote *Details of Convection->Definition->Film Coefficient* na 30 W/(m^{2}.^{0}C)



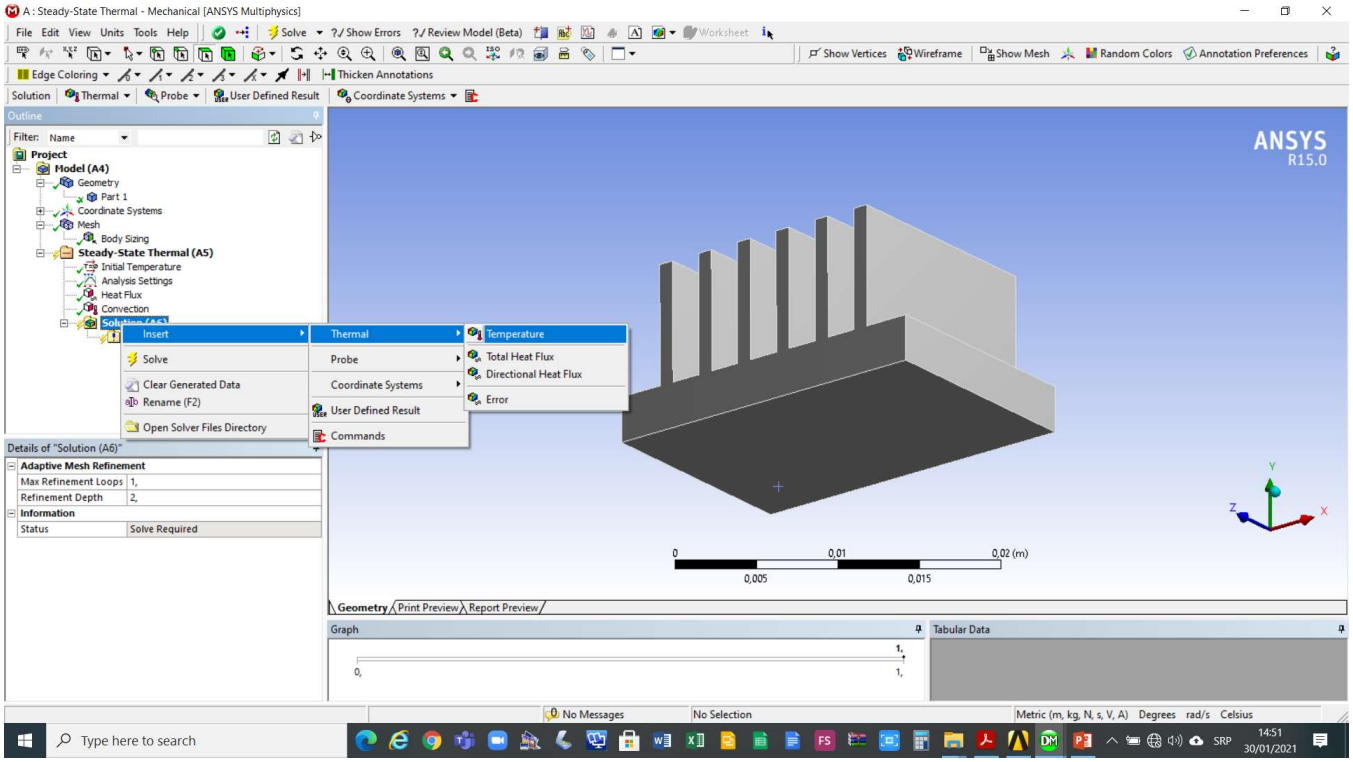
Toplotno opterećenje

Podešavanje temperature vazduha *Details of Convection->Definition->Ambient Temperature* na 28 ^{0}C



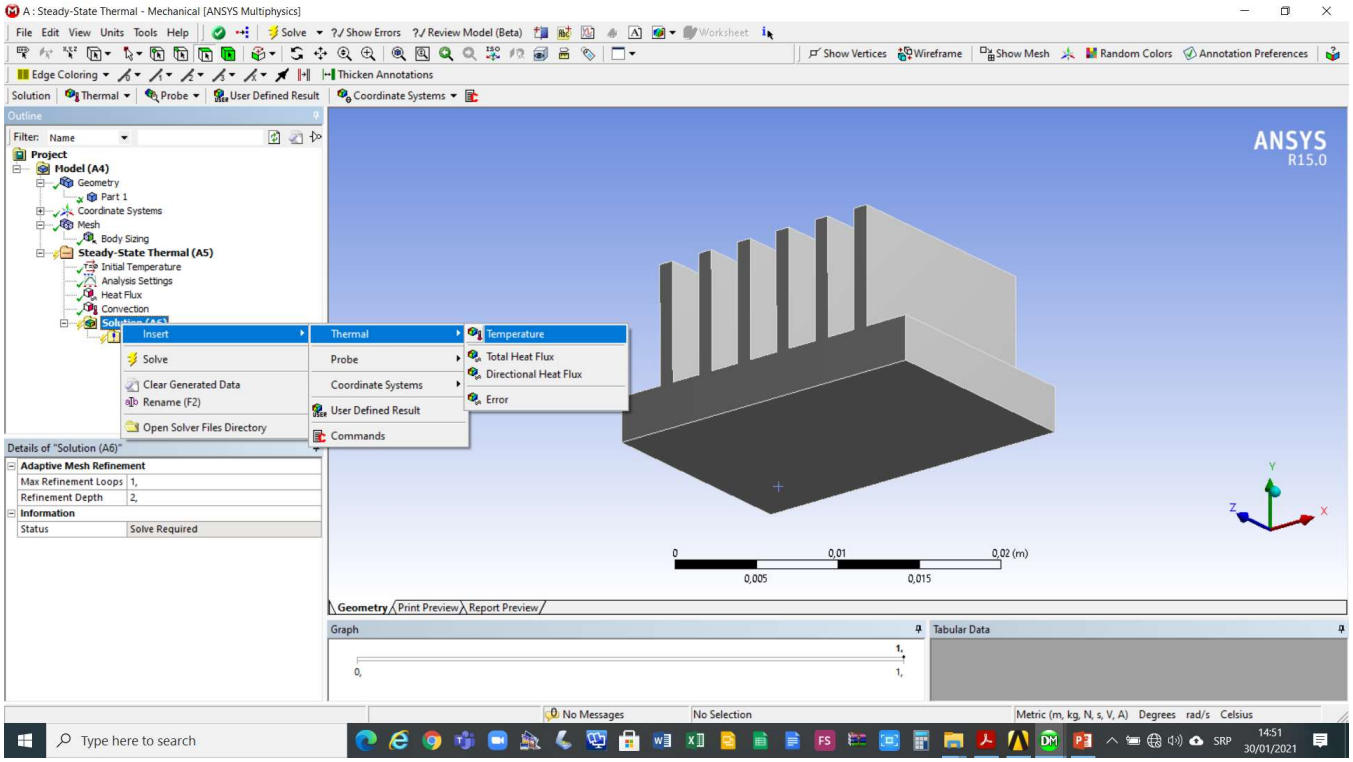
Toplotno opterećenje

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



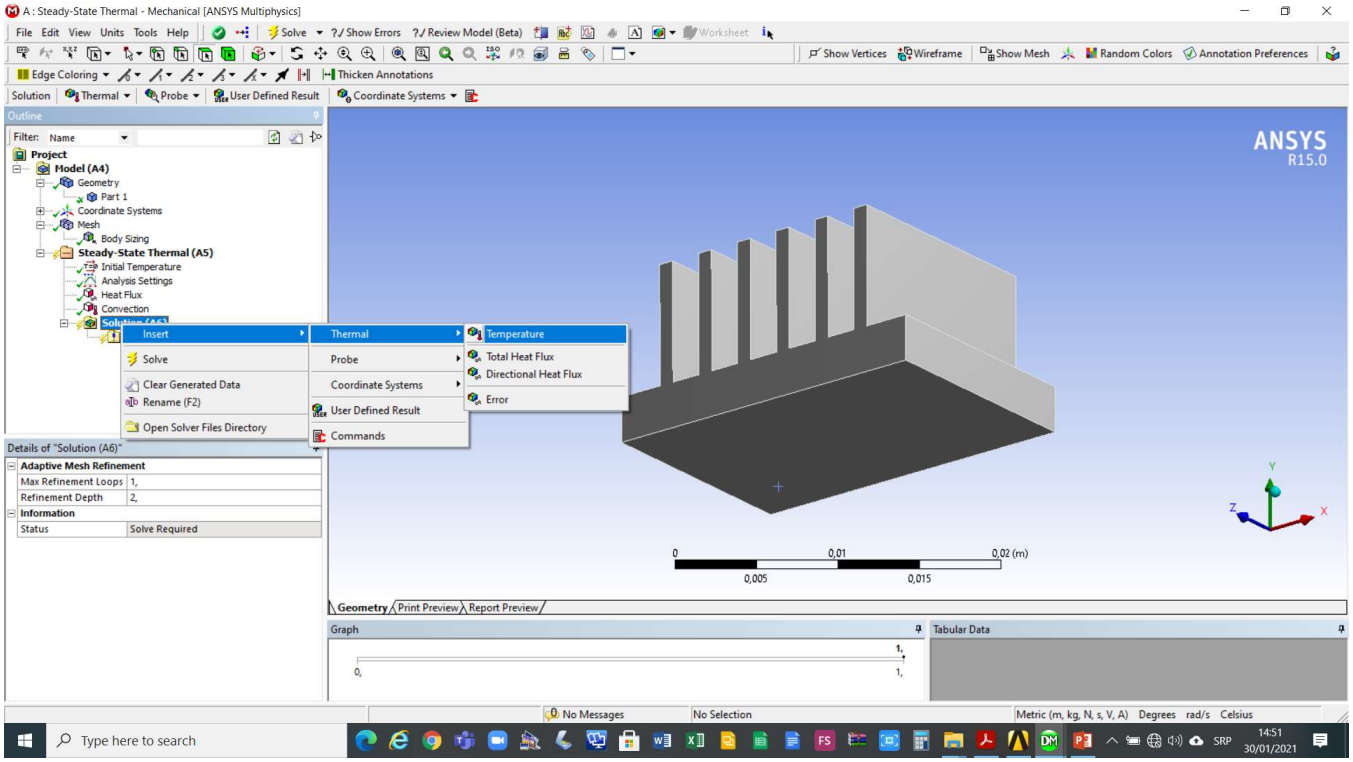
Toplotno opterećenje

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



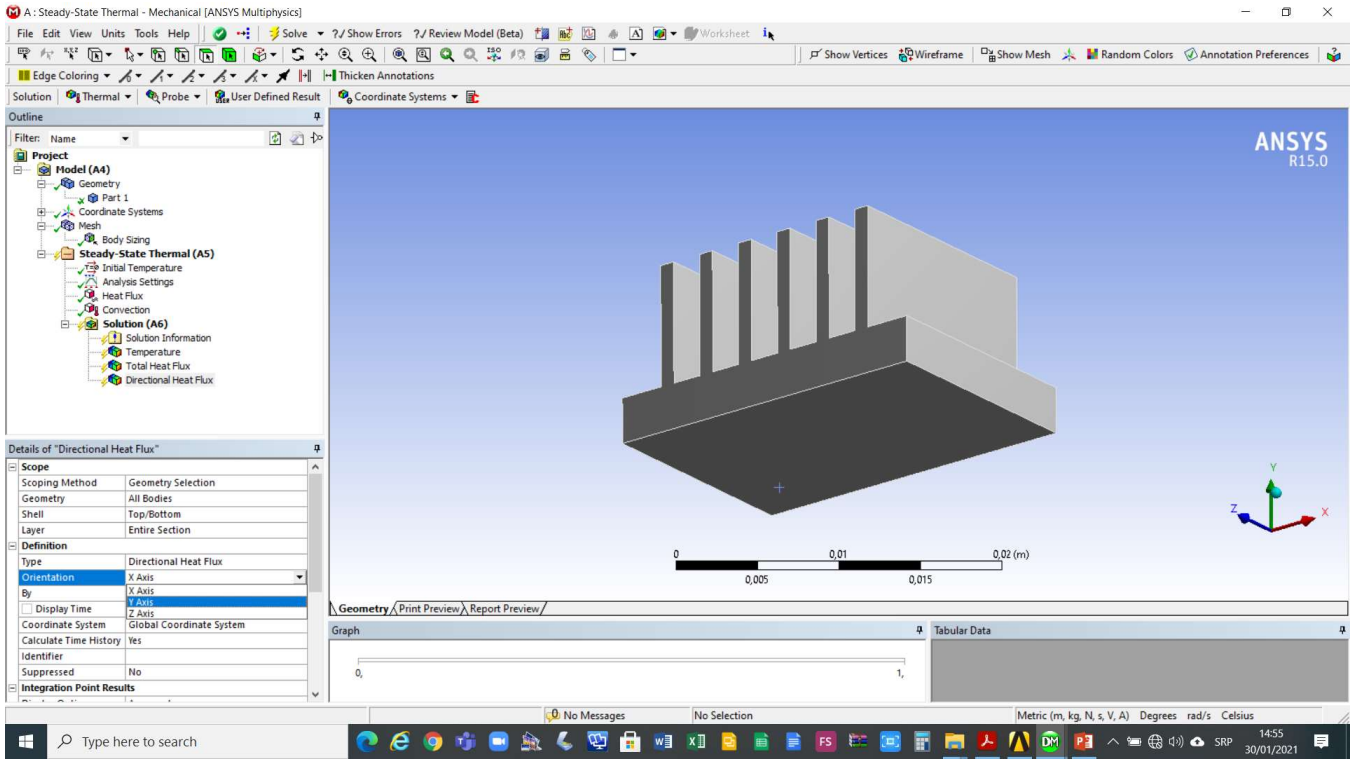
Toplotno opterećenje

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



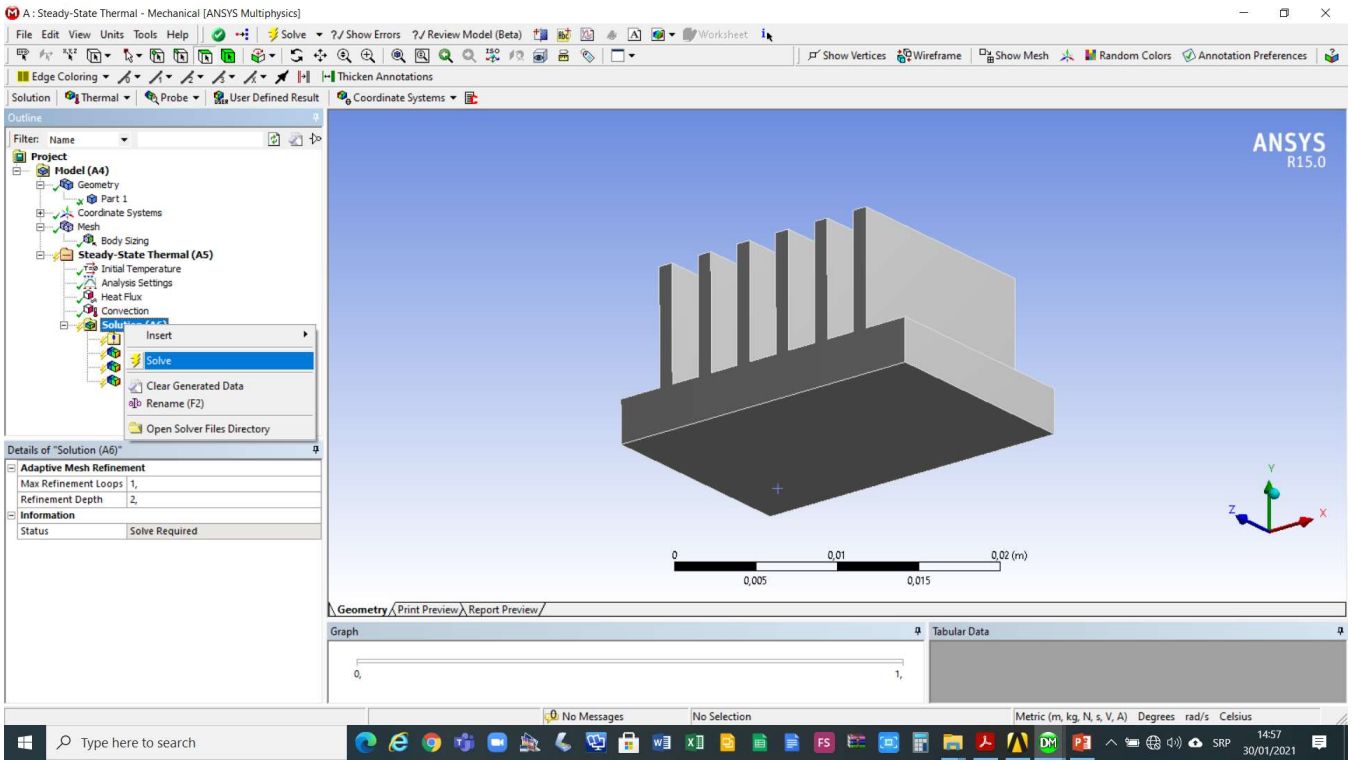
Toplotno opterećenje

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



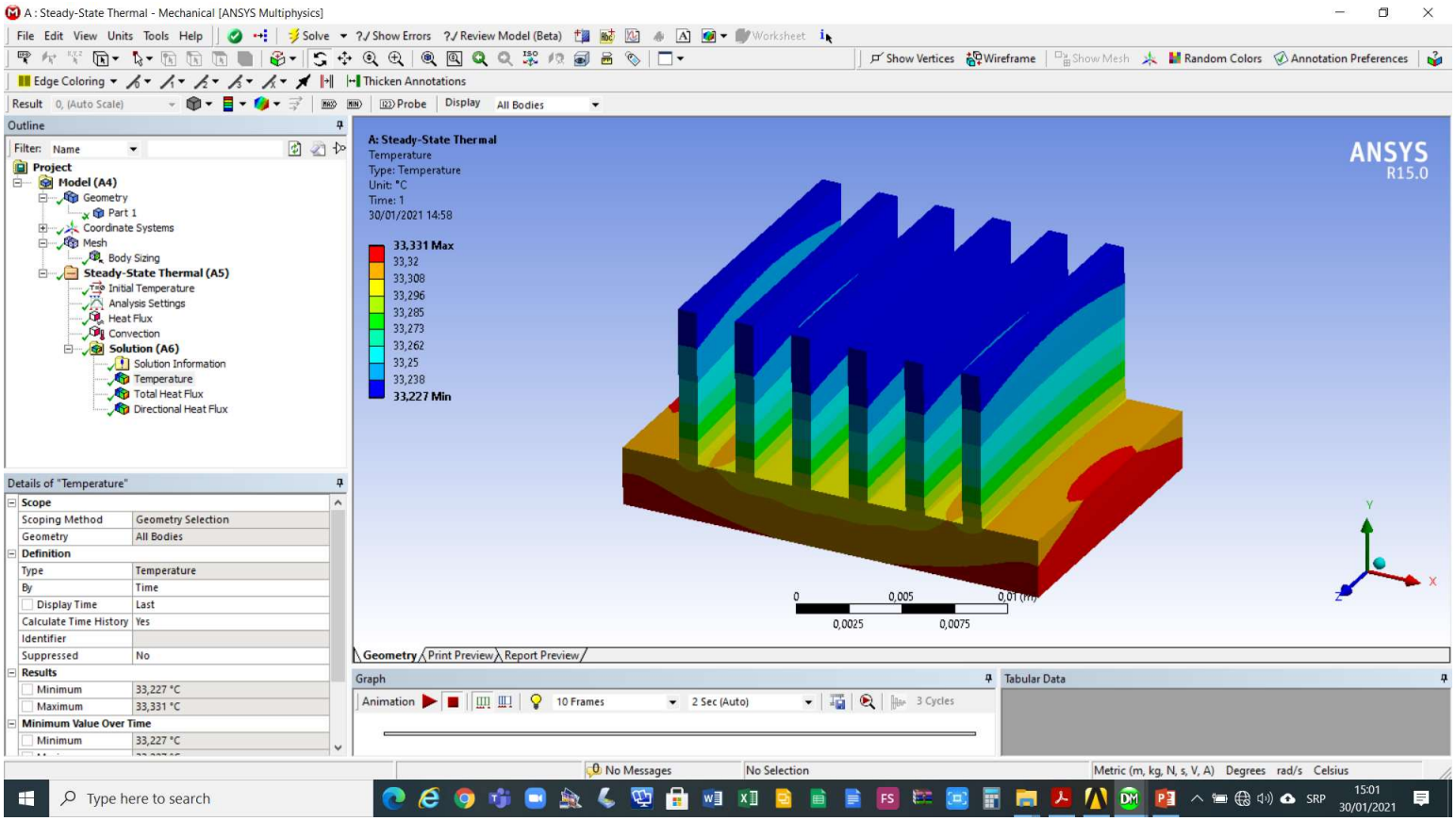
Toplotno opterećenje

Aktivirati izvršenje analize *Solution->Solve*



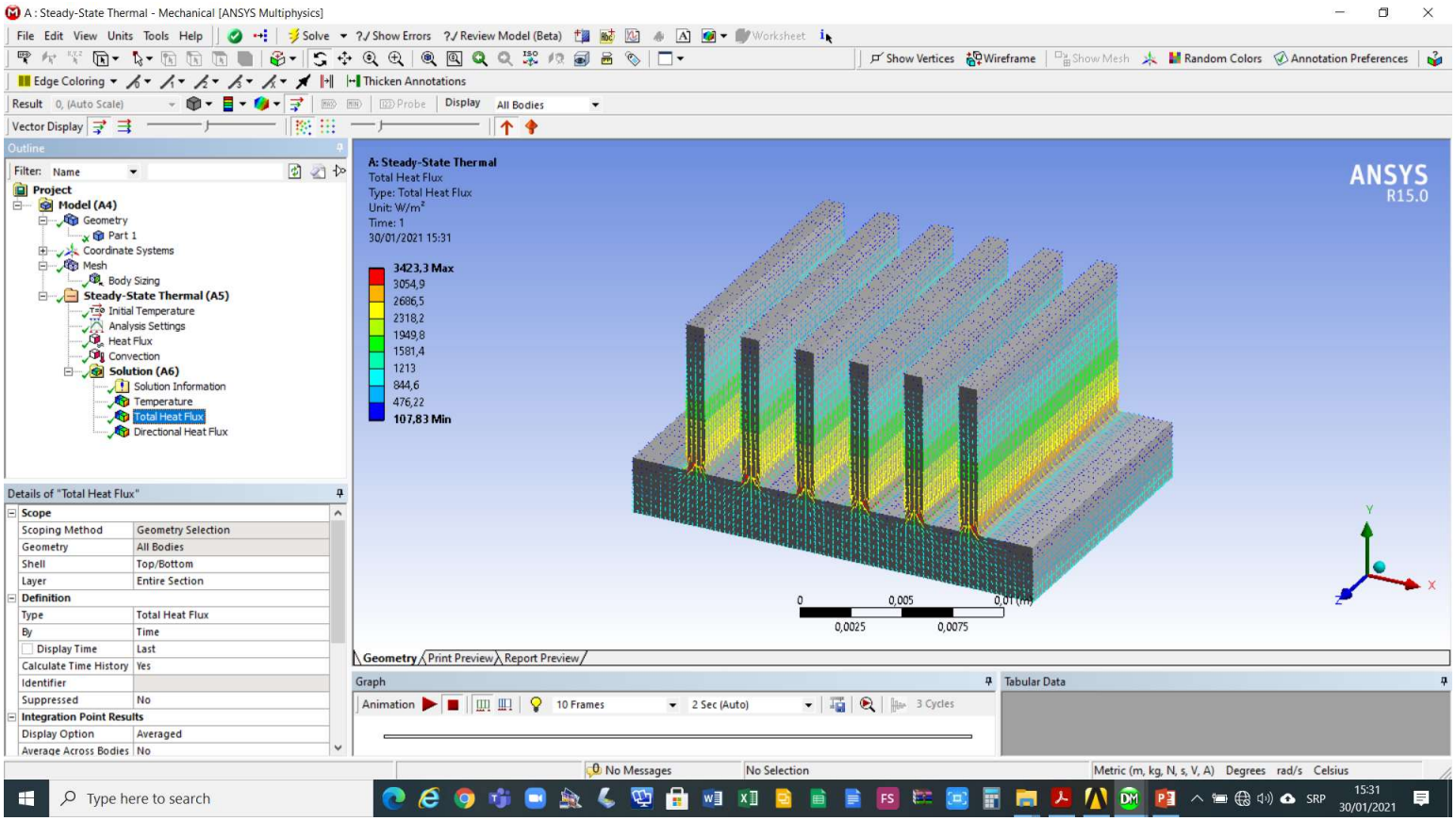
Toplotno opterećenje

Raspodjela temperature u ravnotežnom stanju



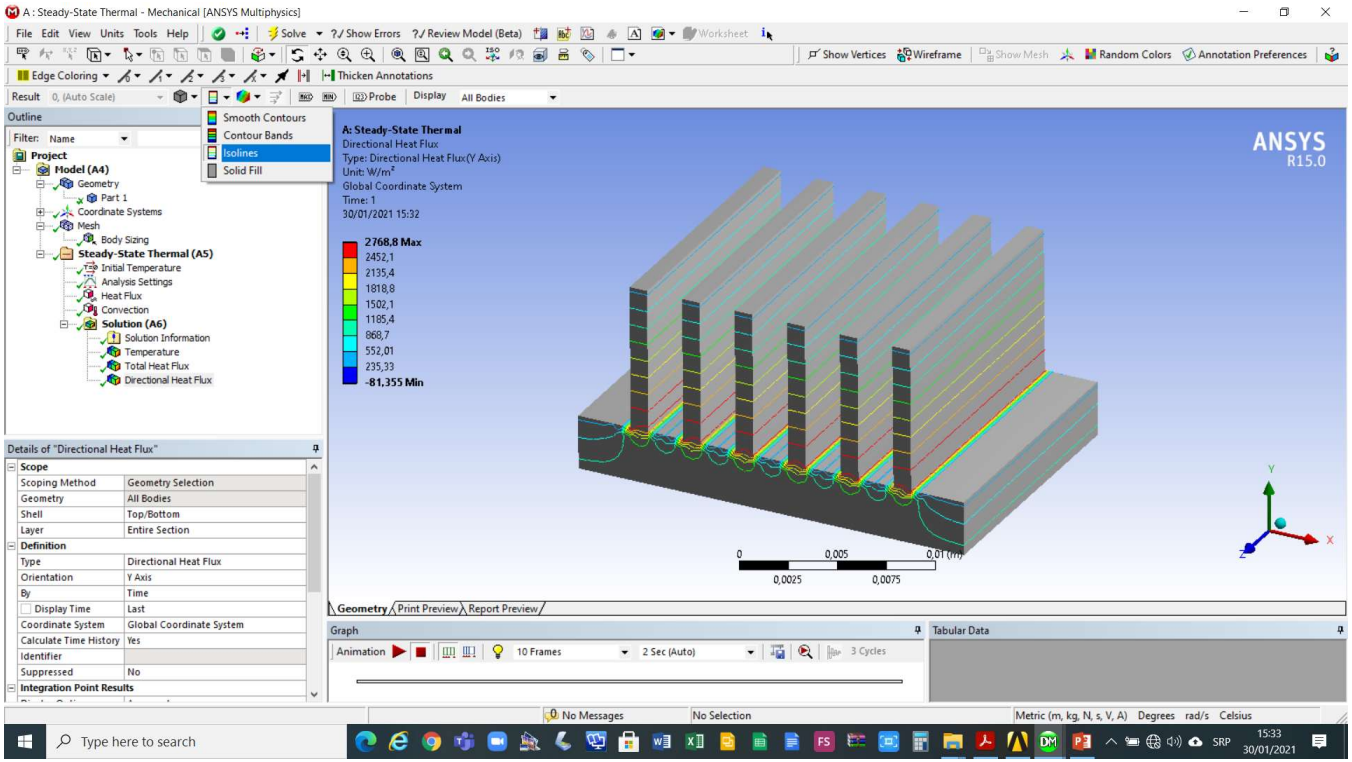
Toplotno opterećenje

Raspodjela toplotnog fluksa



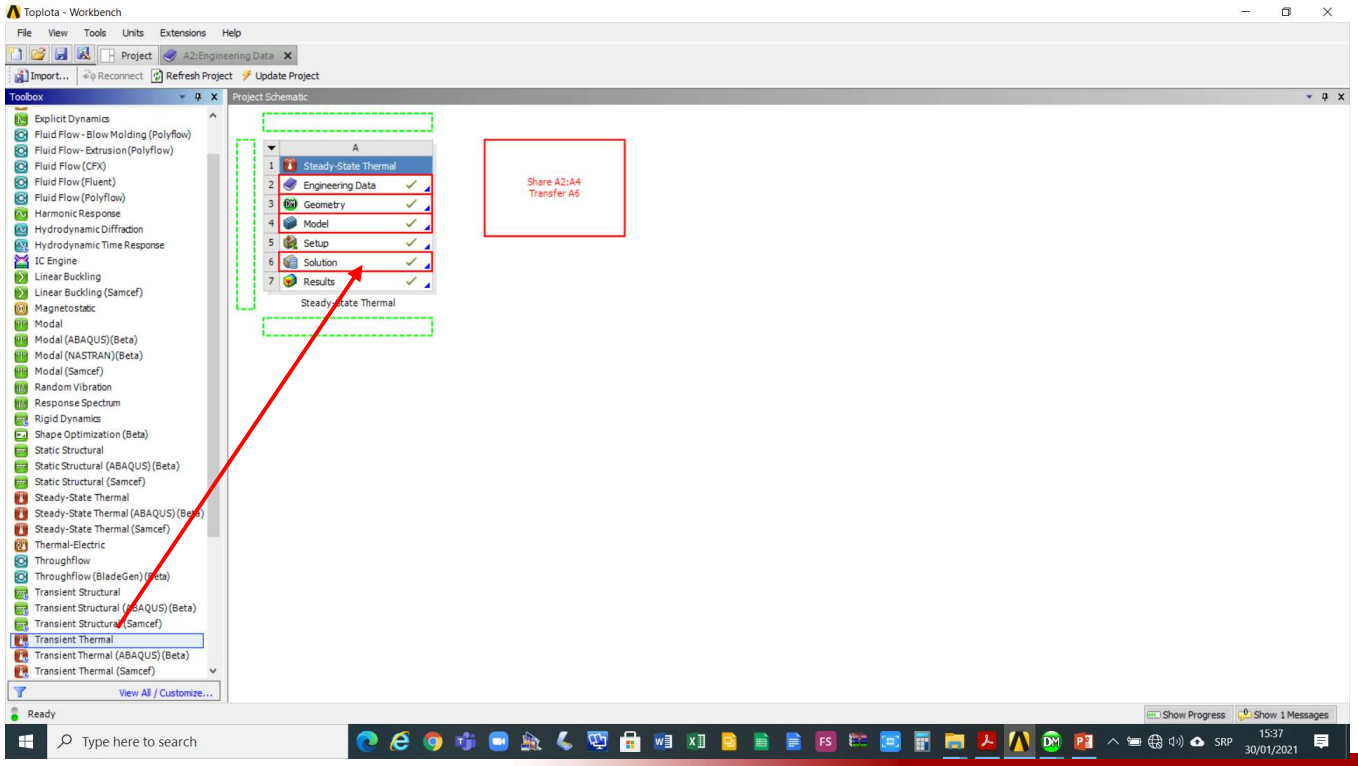
Toplotno opterećenje

Raspodjela toplotnog fluksa u pavcu normalnom na donju površinu



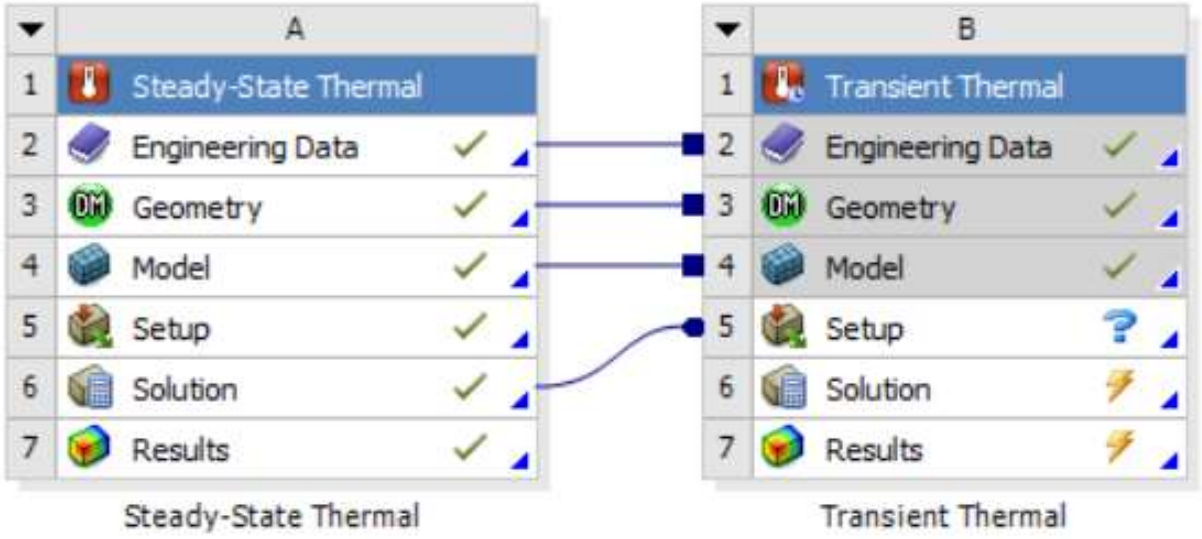
Toplotno opterećenje

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



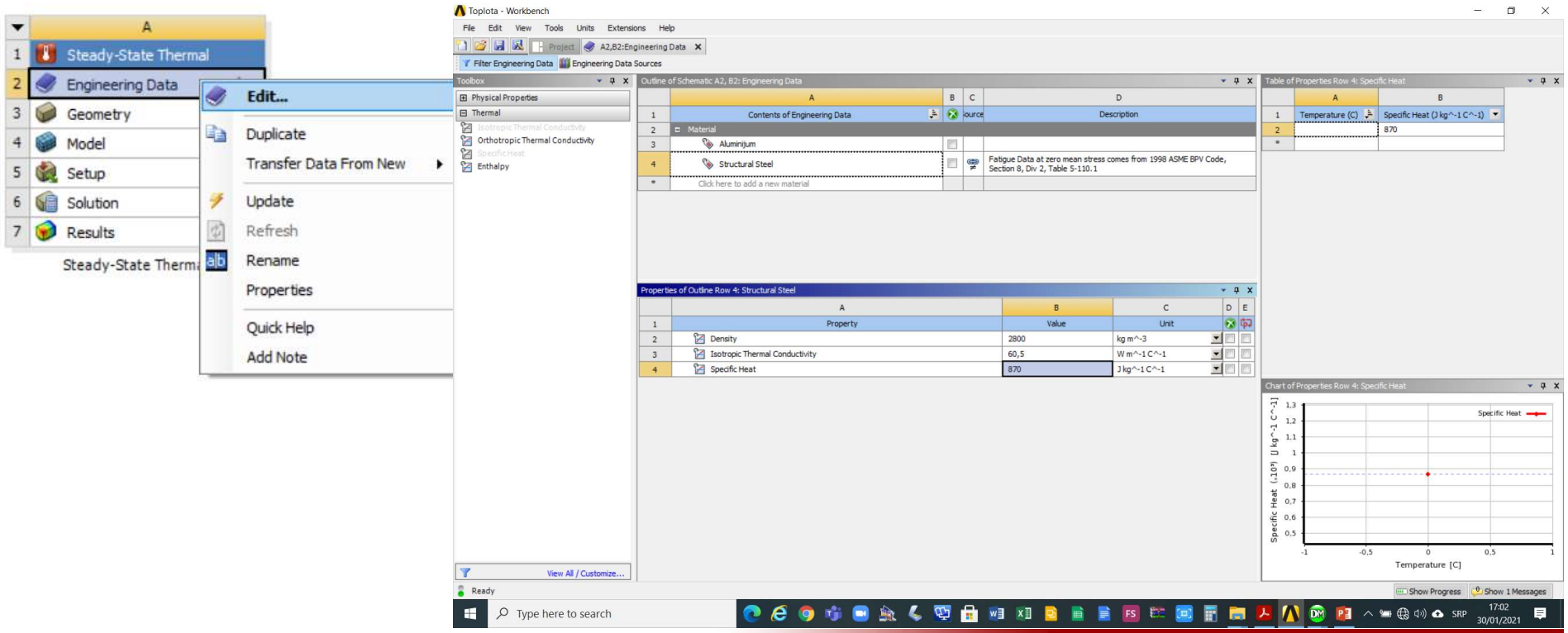
Toplotno opterećenje

Prethodni postupak omogućava potrebno povezivanje stacionarne i netacionarne termalne analize



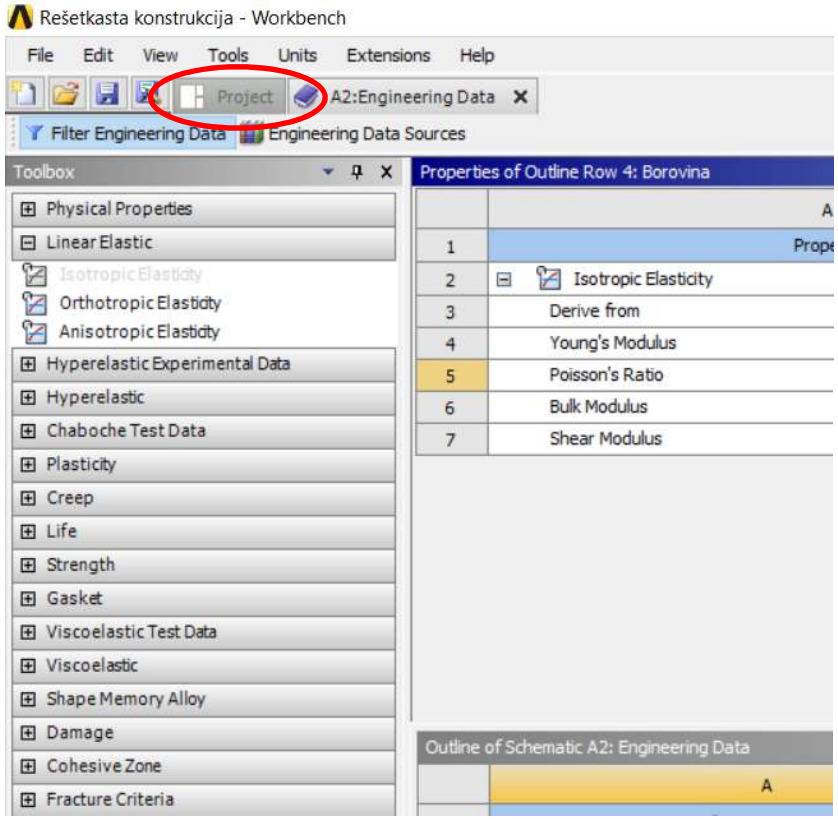
Toplotno opterećenje

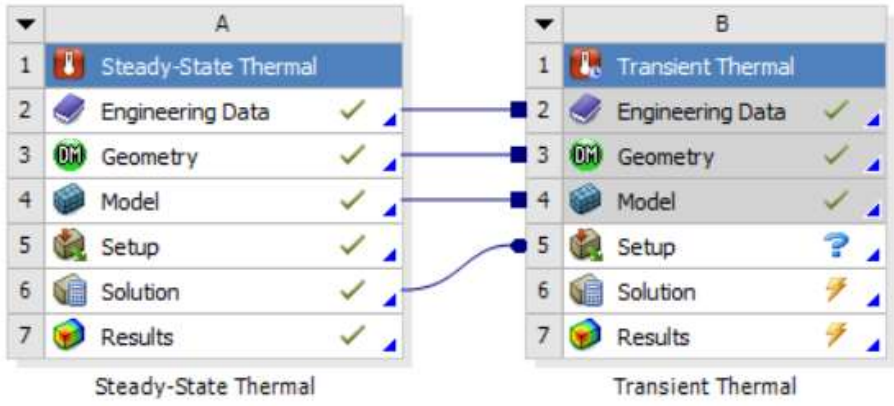
Aktivirati modul Engineering Data (*Engineering Data->Edit*) i dodati podatke o gustini i specifičnoj toploti



Toplotno opterećenje

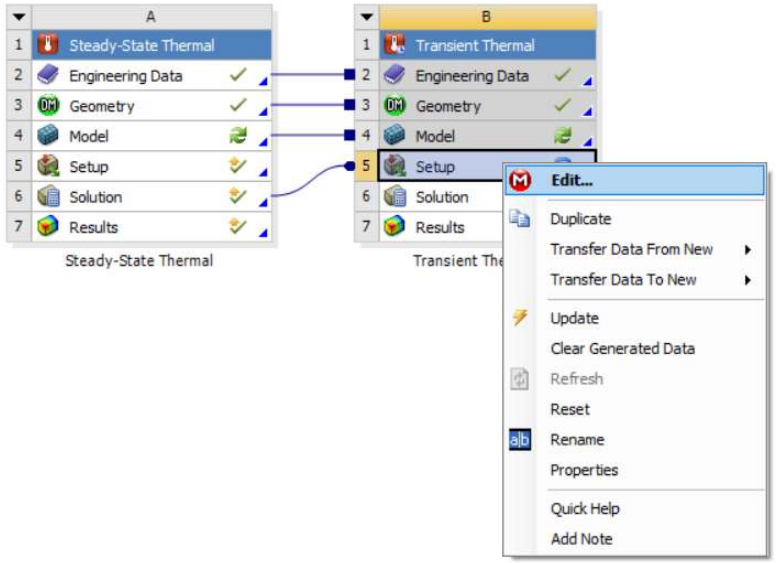
Izabrati opciju *Project* za povratak na shemu projekta

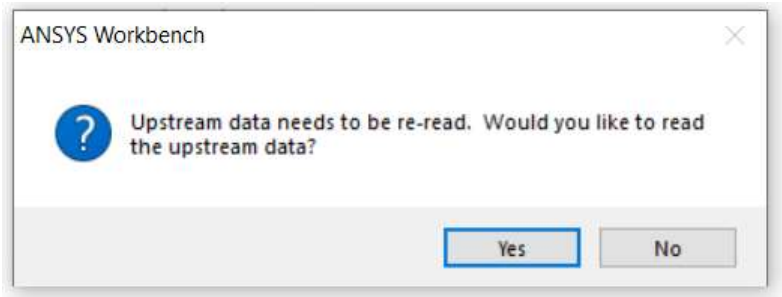




Toplotno opterećenje

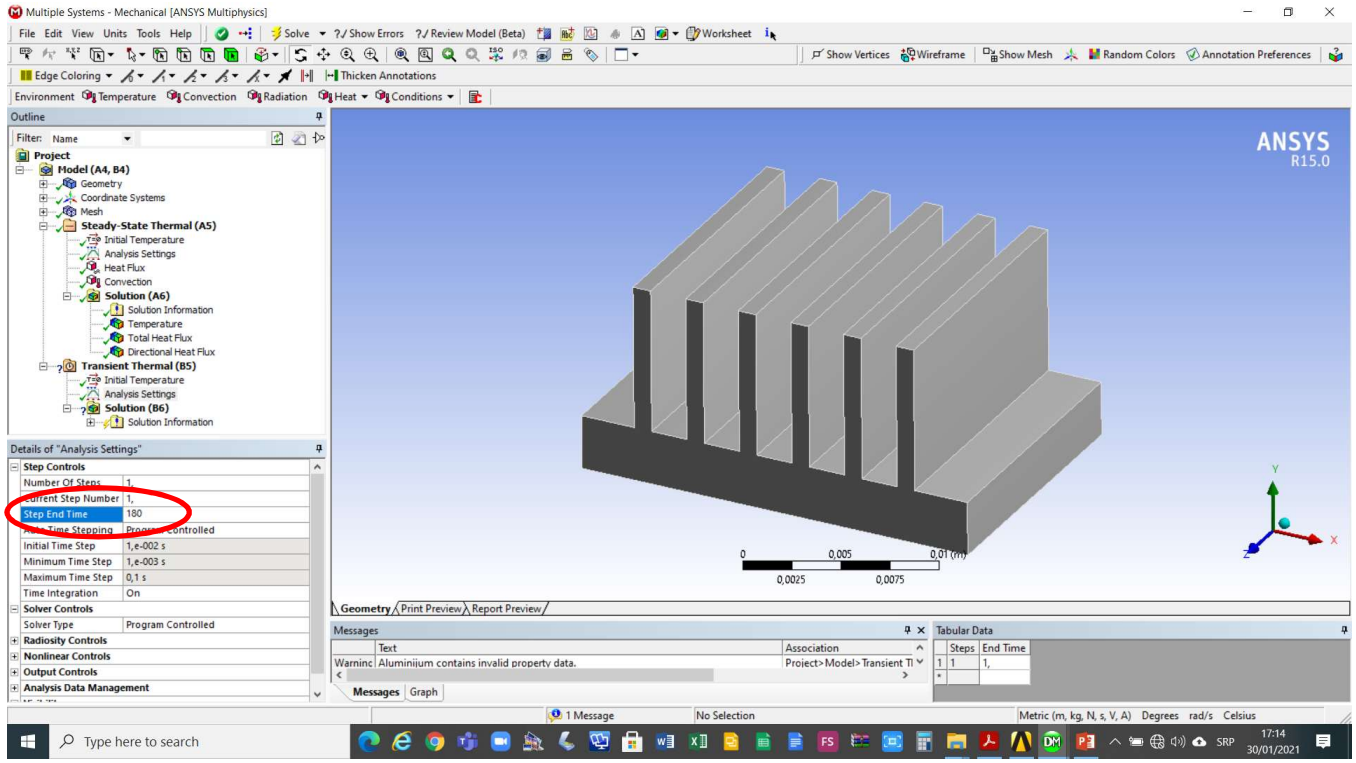
Aktivirati modul Multiple Systems (*Transient Thermal->Setup->Edit*) i odgovoriti sa *Yes* kako bi se učitali novounešeni podaci





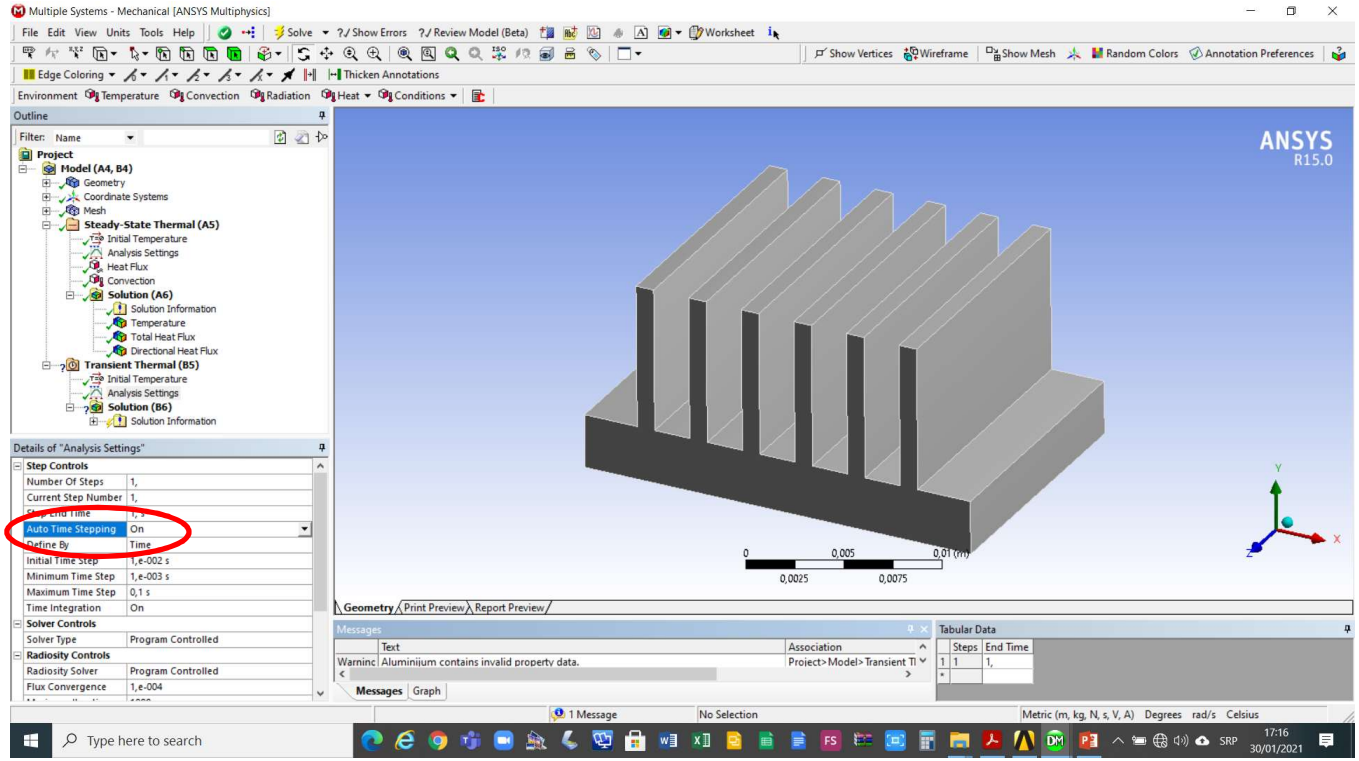
Toplotno opterećenje

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



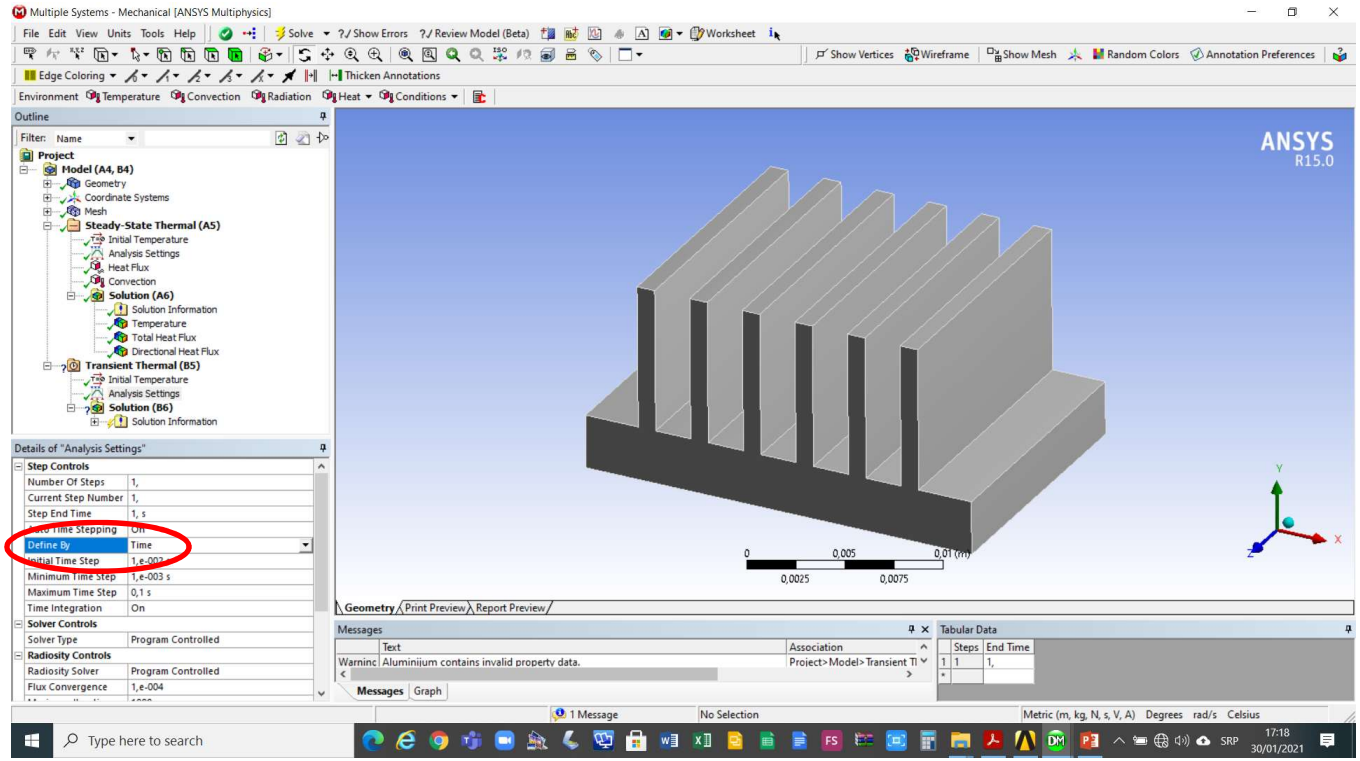
Toplotno opterećenje

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



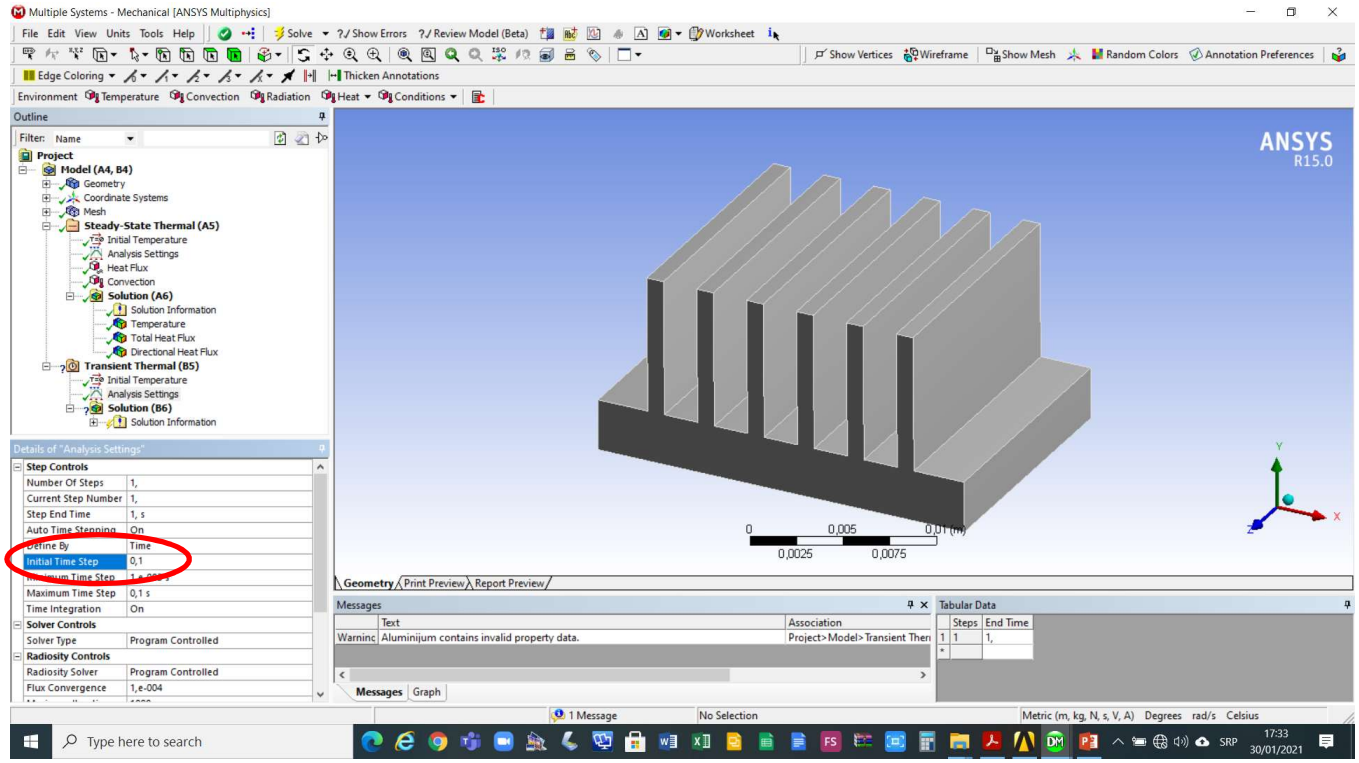
Toplotno opterećenje

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



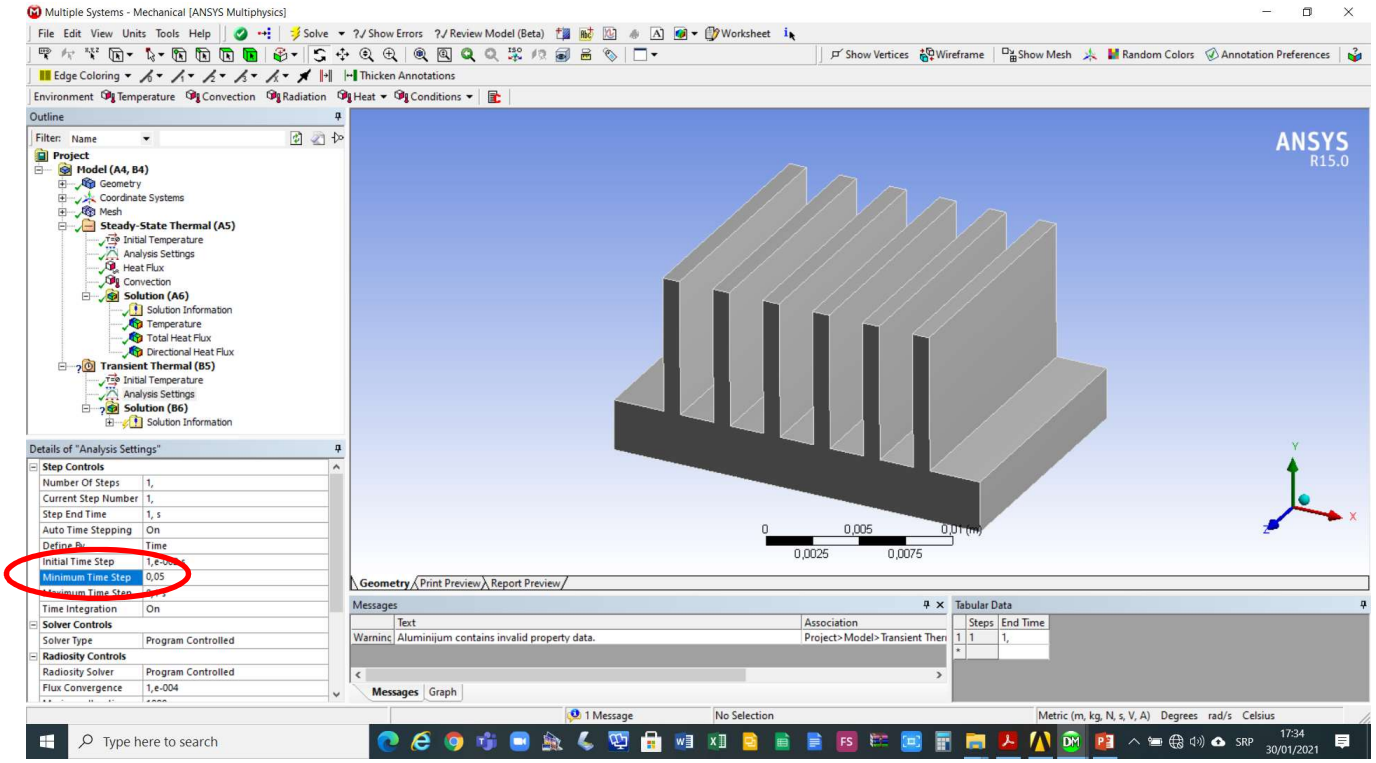
Toplotno opterećenje

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



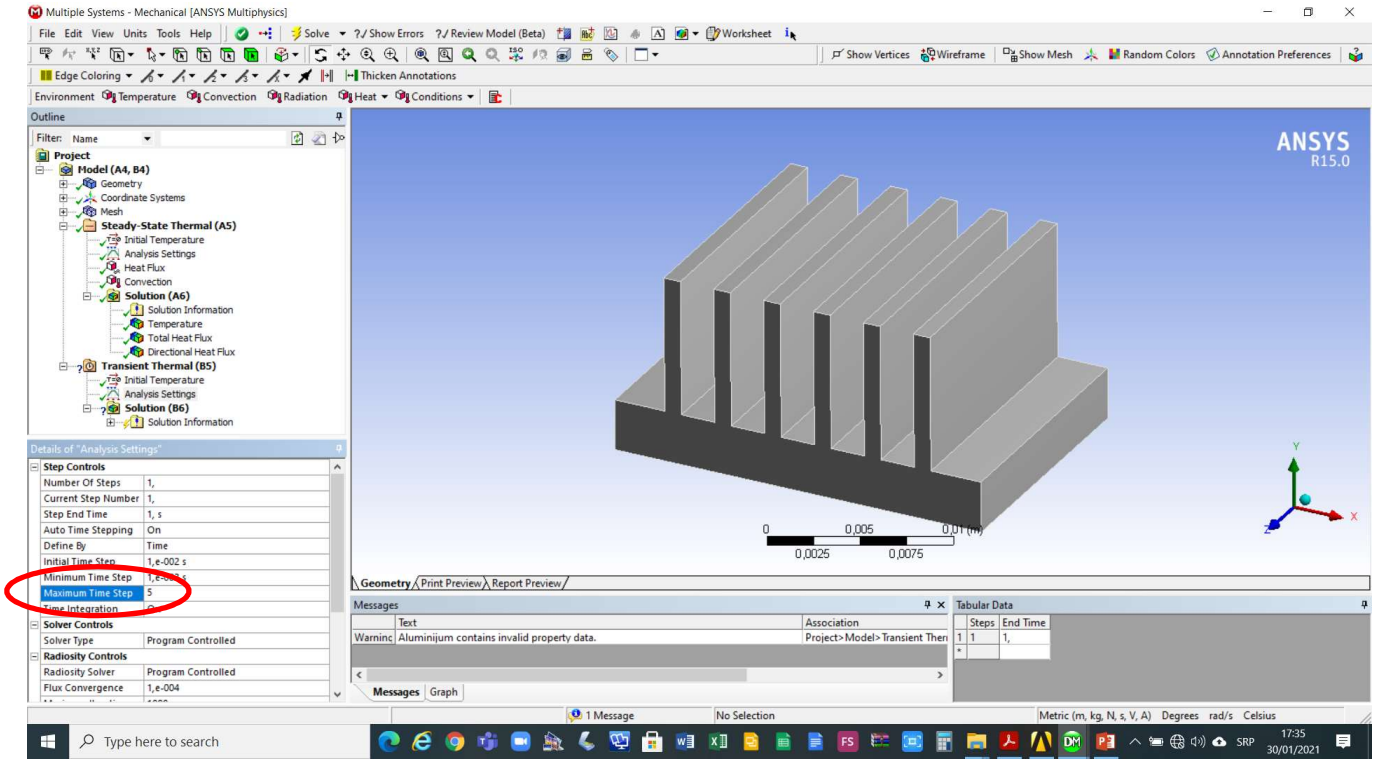
Toplotno opterećenje

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

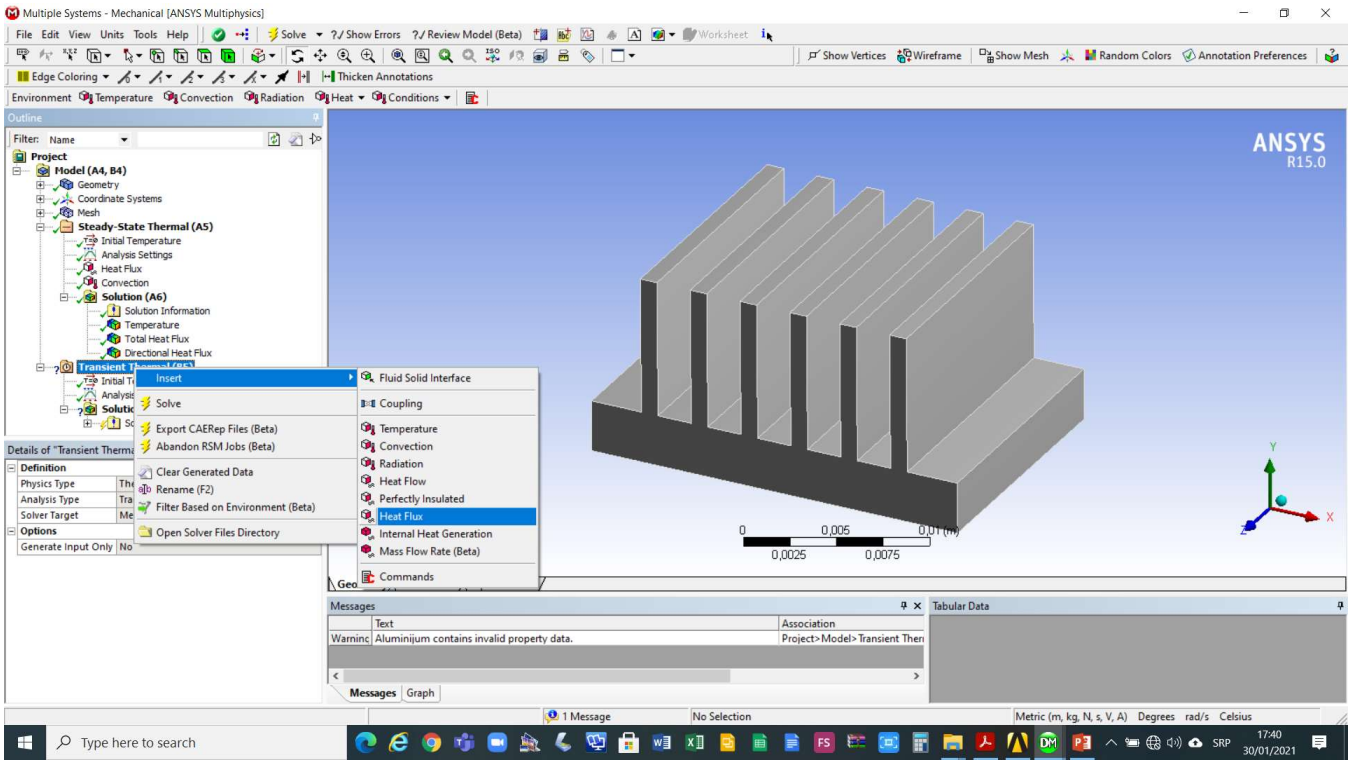


Toplotno opterećenje

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

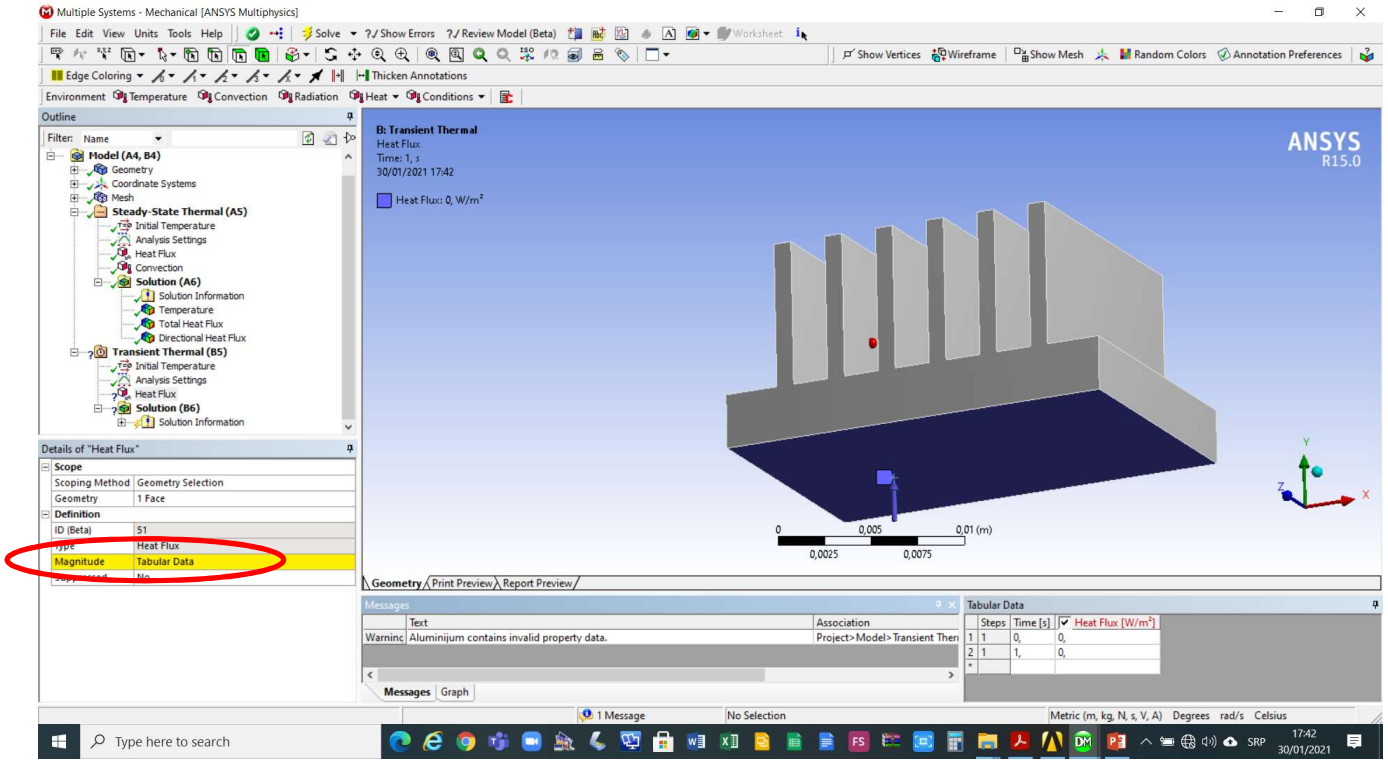
Toplotno opterećenje

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



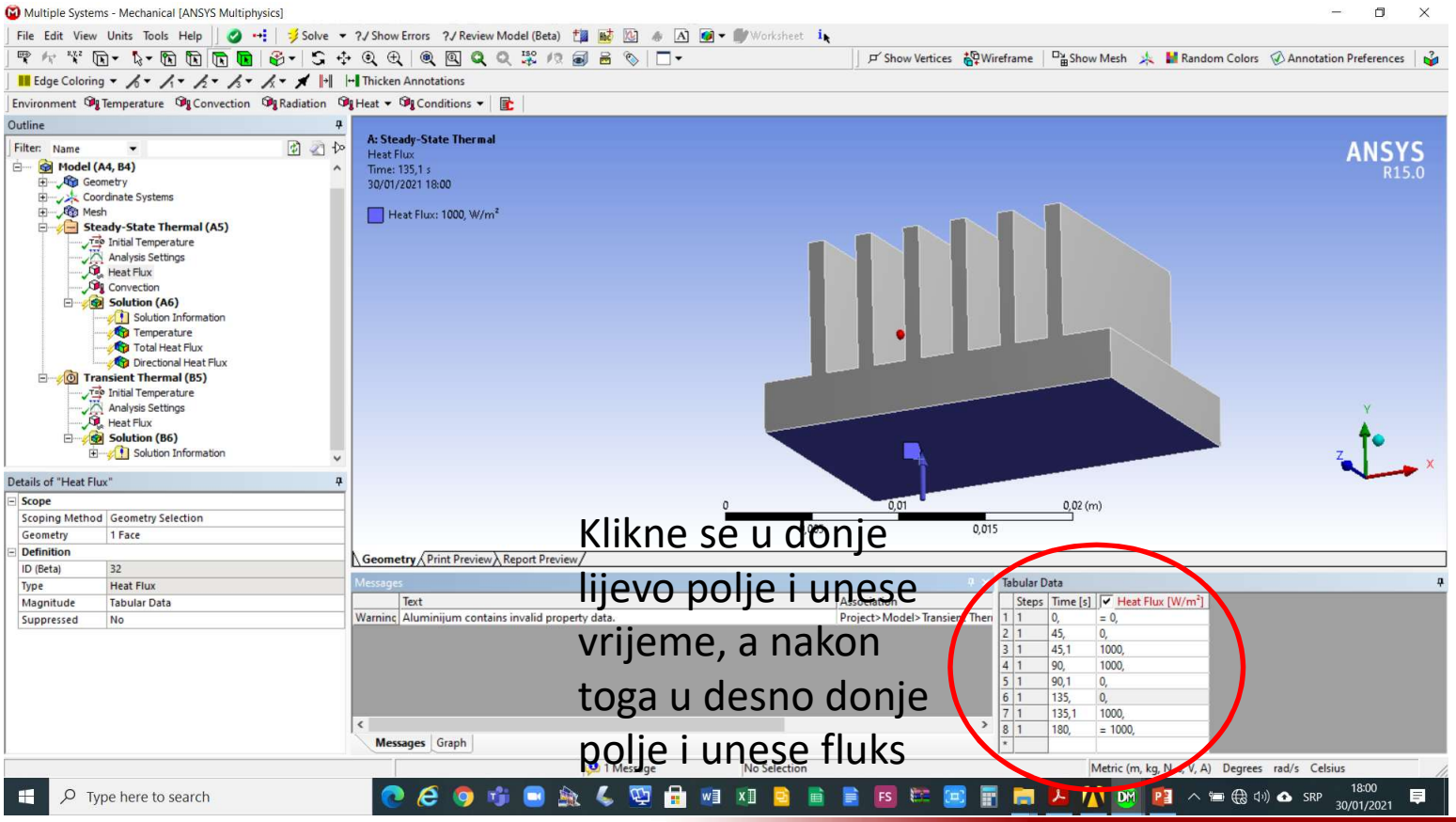
Toplotno opterećenje

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



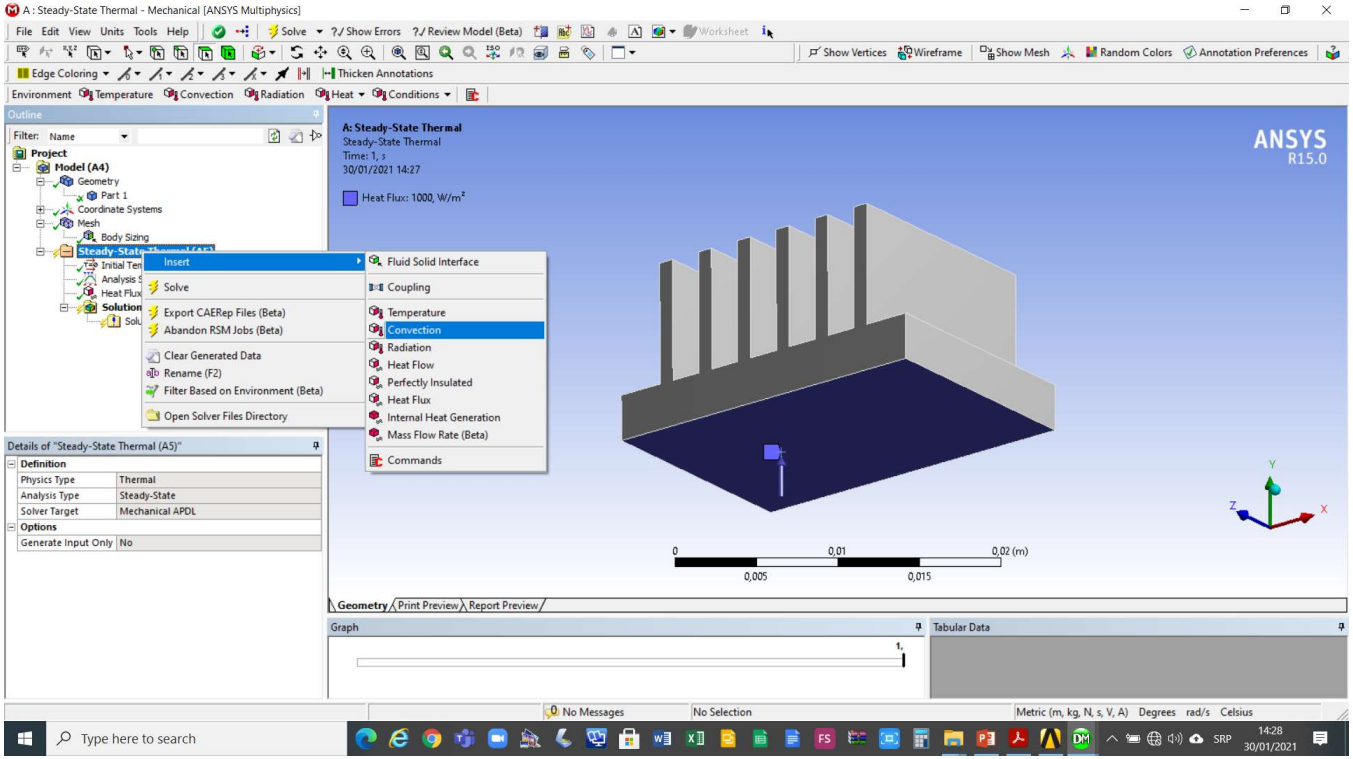
Toplotno opterećenje

Unijeti tabelarne podatke o toplotnom fluksu kao na slici



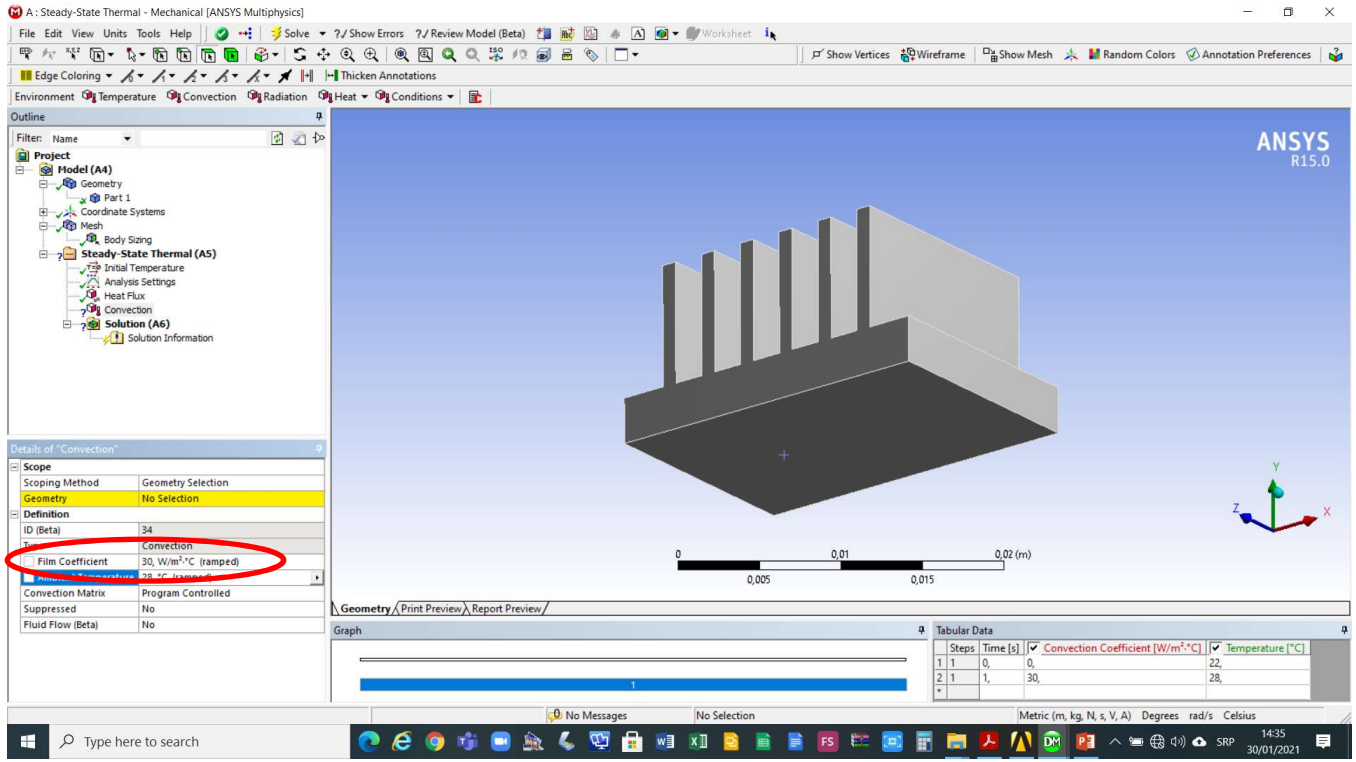
Toplotno opterećenje

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



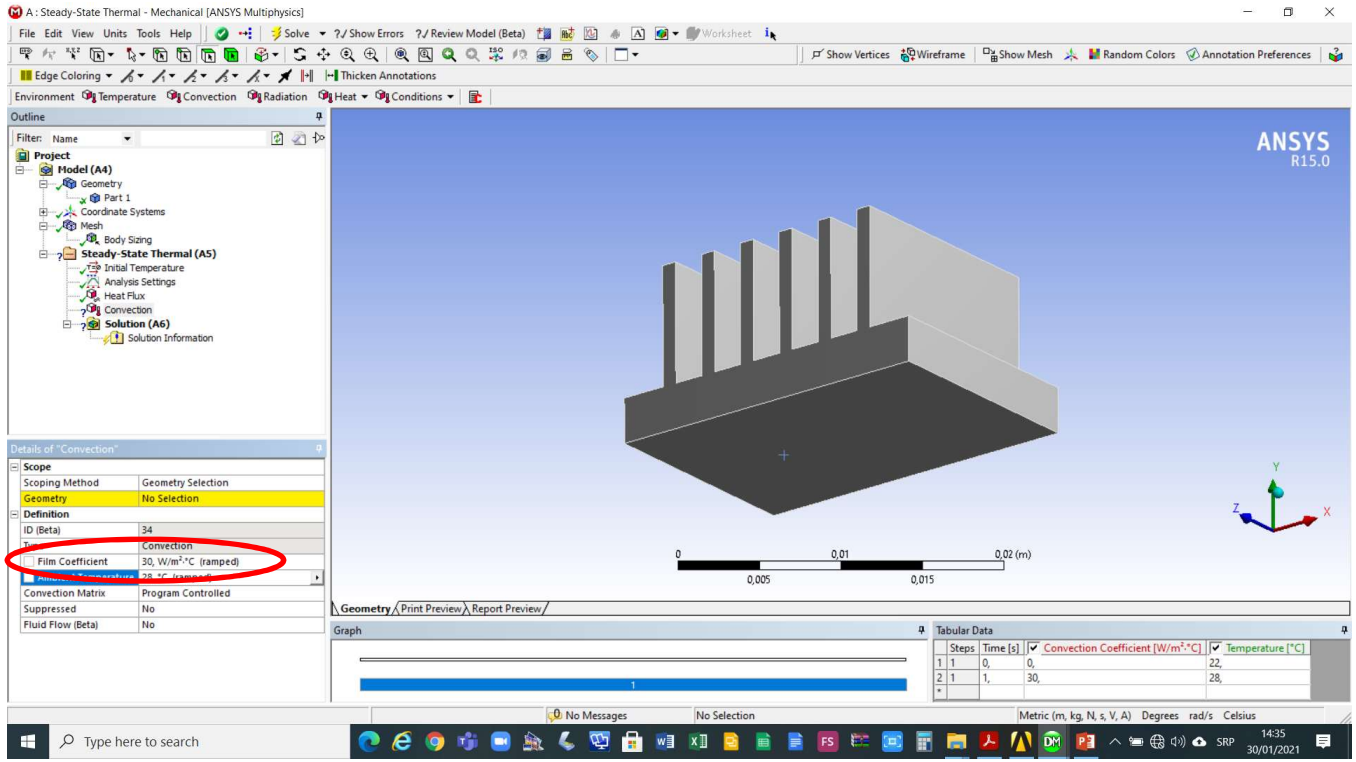
Toplotno opterećenje

Podešavanje koeficijenta prenosa toplote *Details of Convection->Definition->Film Coefficient* na 30 W/(m^{2}.^{0}C)



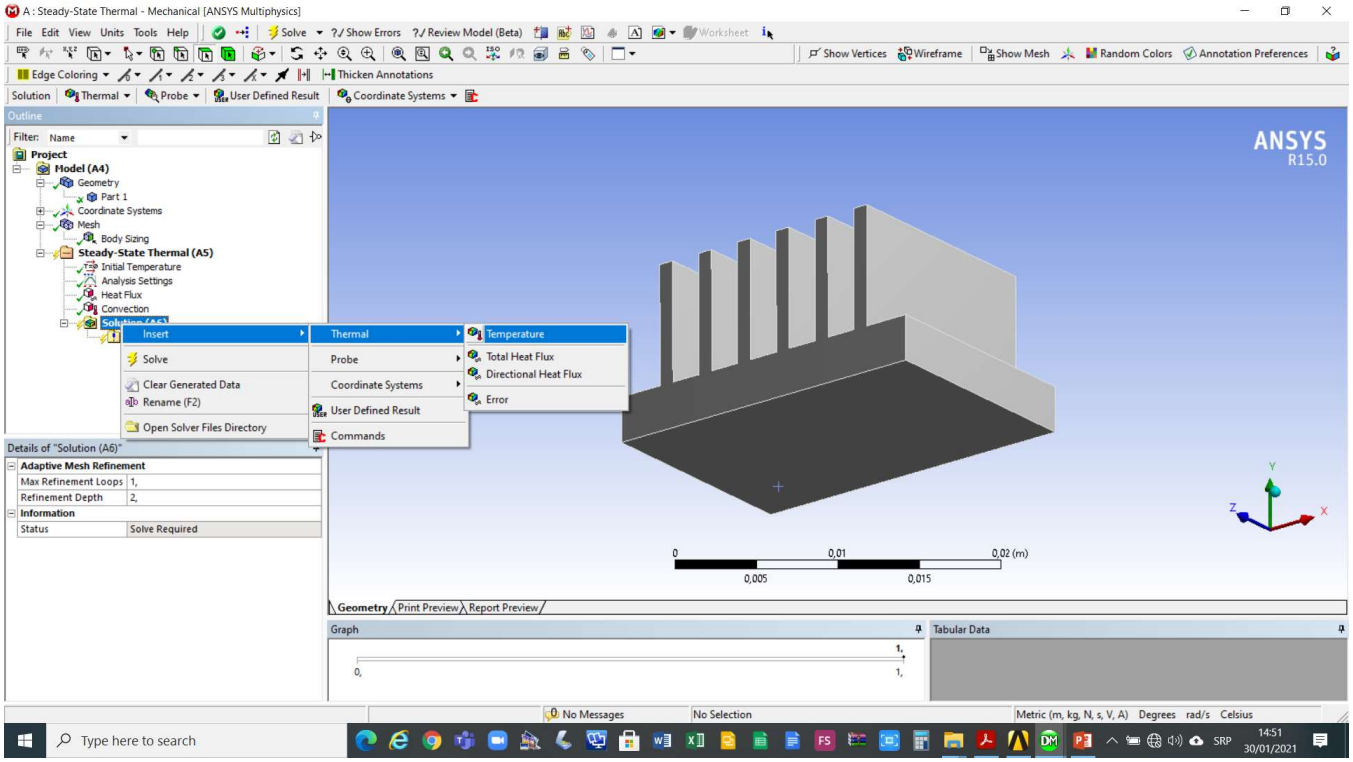
Toplotno opterećenje

Podešavanje temperature vazduha *Details of Convection->Definition->Ambient Temperature* na 28 ^{0}C



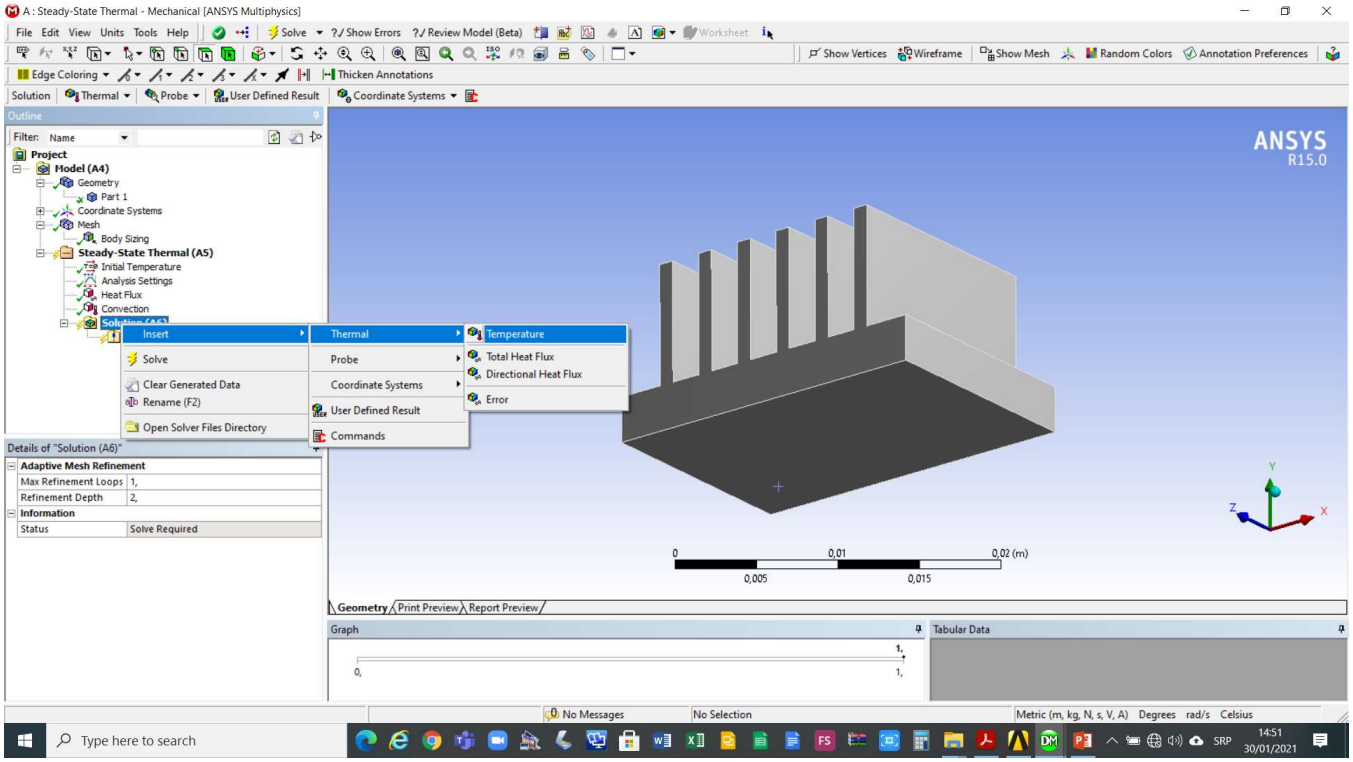
Toplotno opterećenje

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



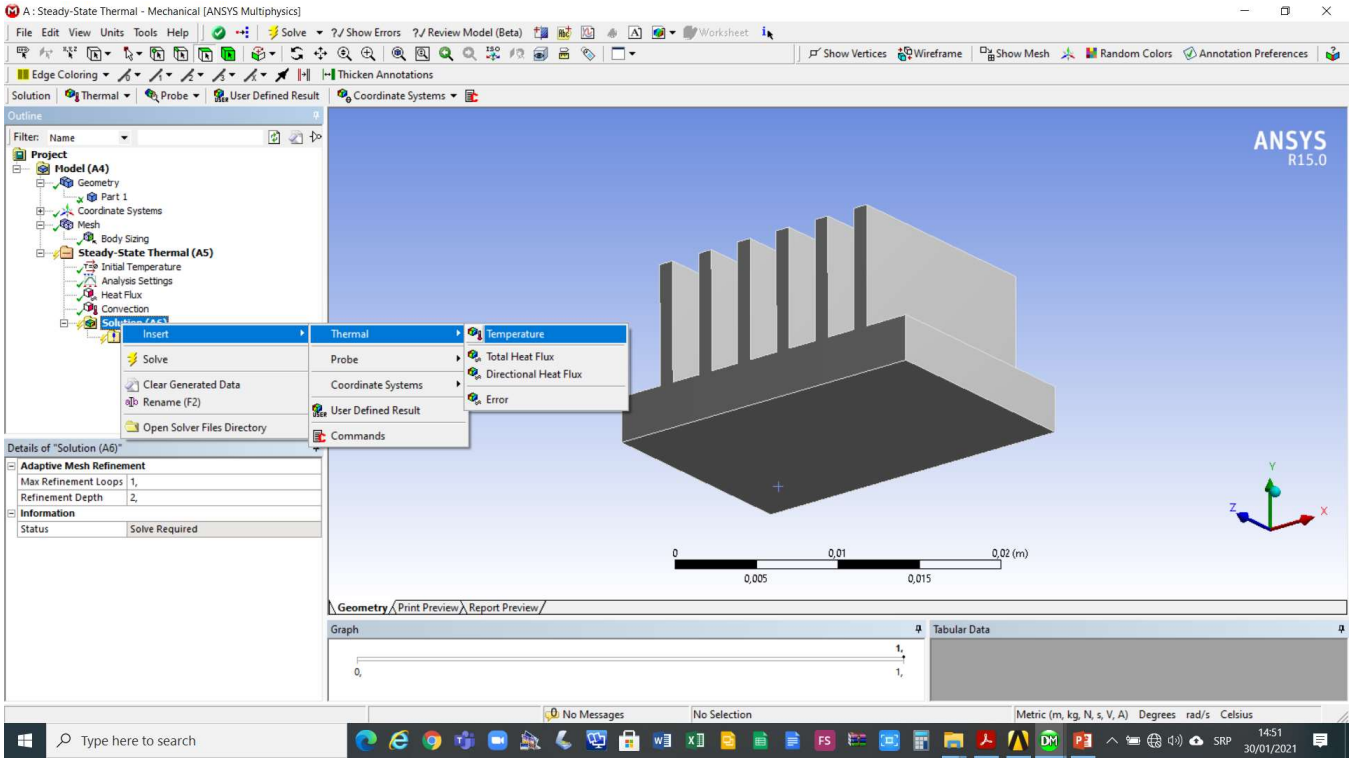
Toplotno opterećenje

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



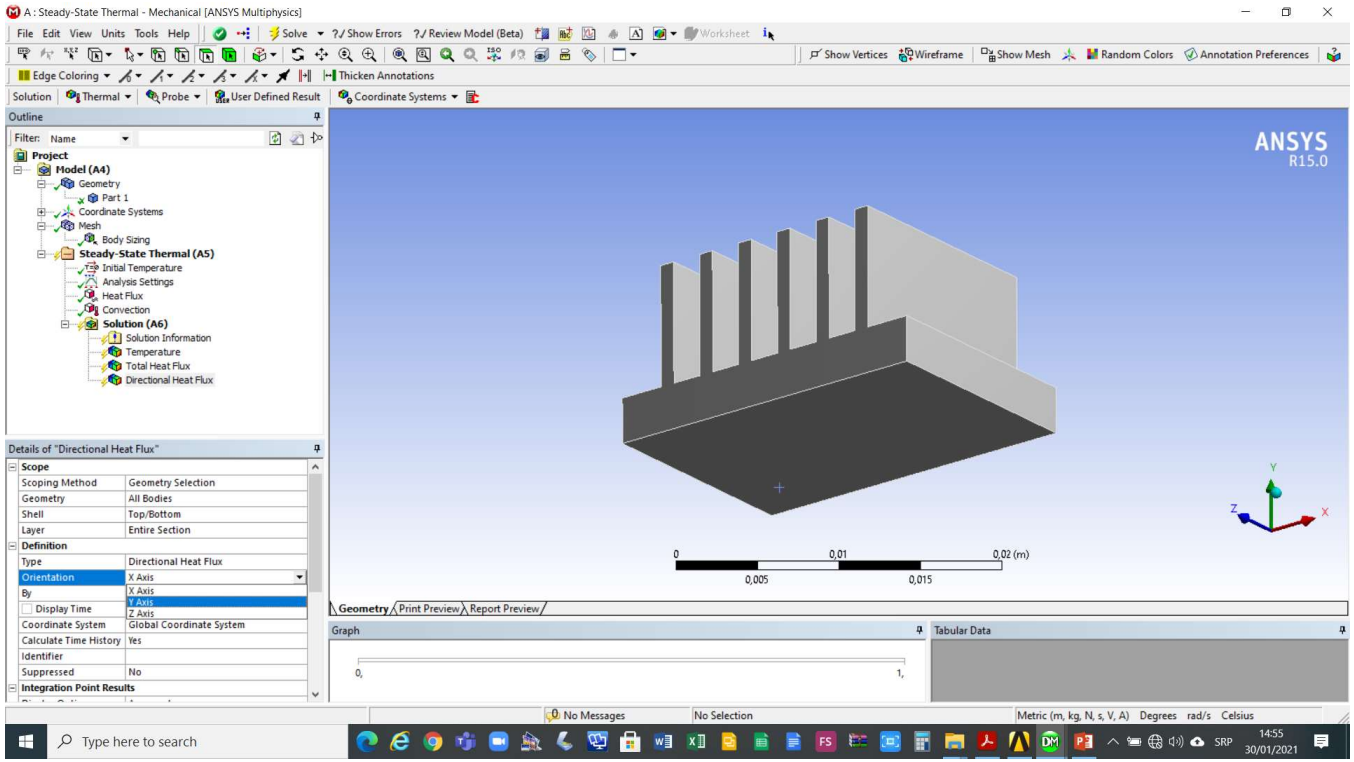
Toplotno opterećenje

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



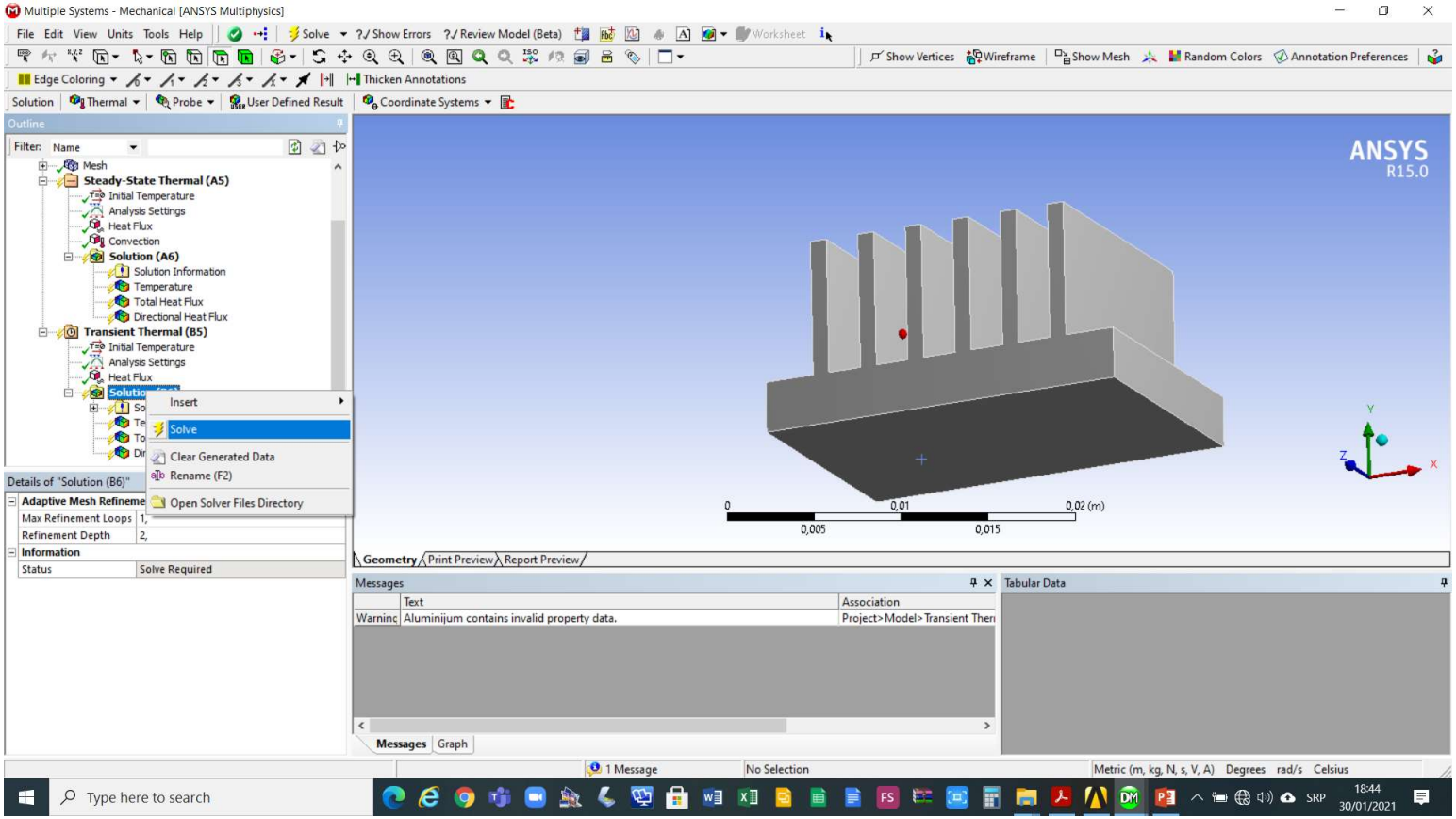
Toplotno opterećenje

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



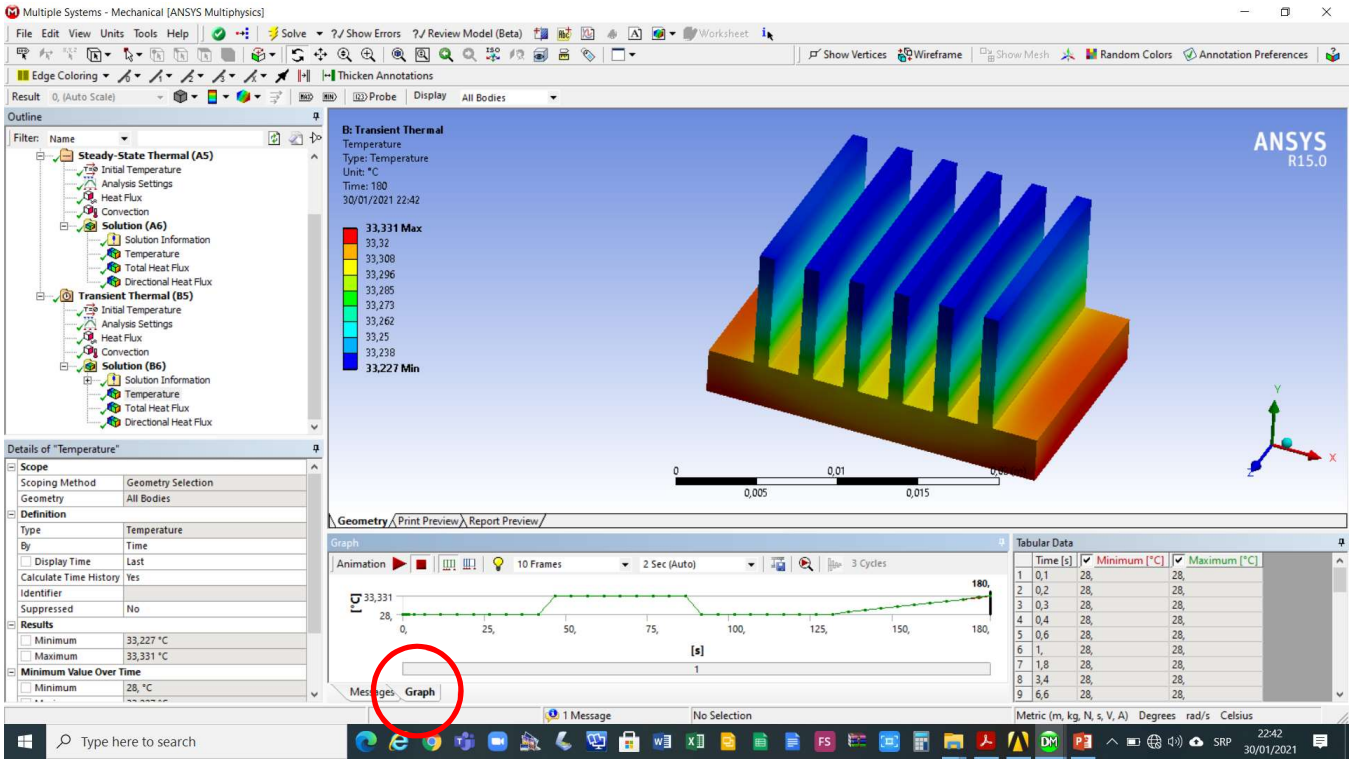
Toplotno opterećenje

Aktivirati izvršenje analize *Solution->Solve*



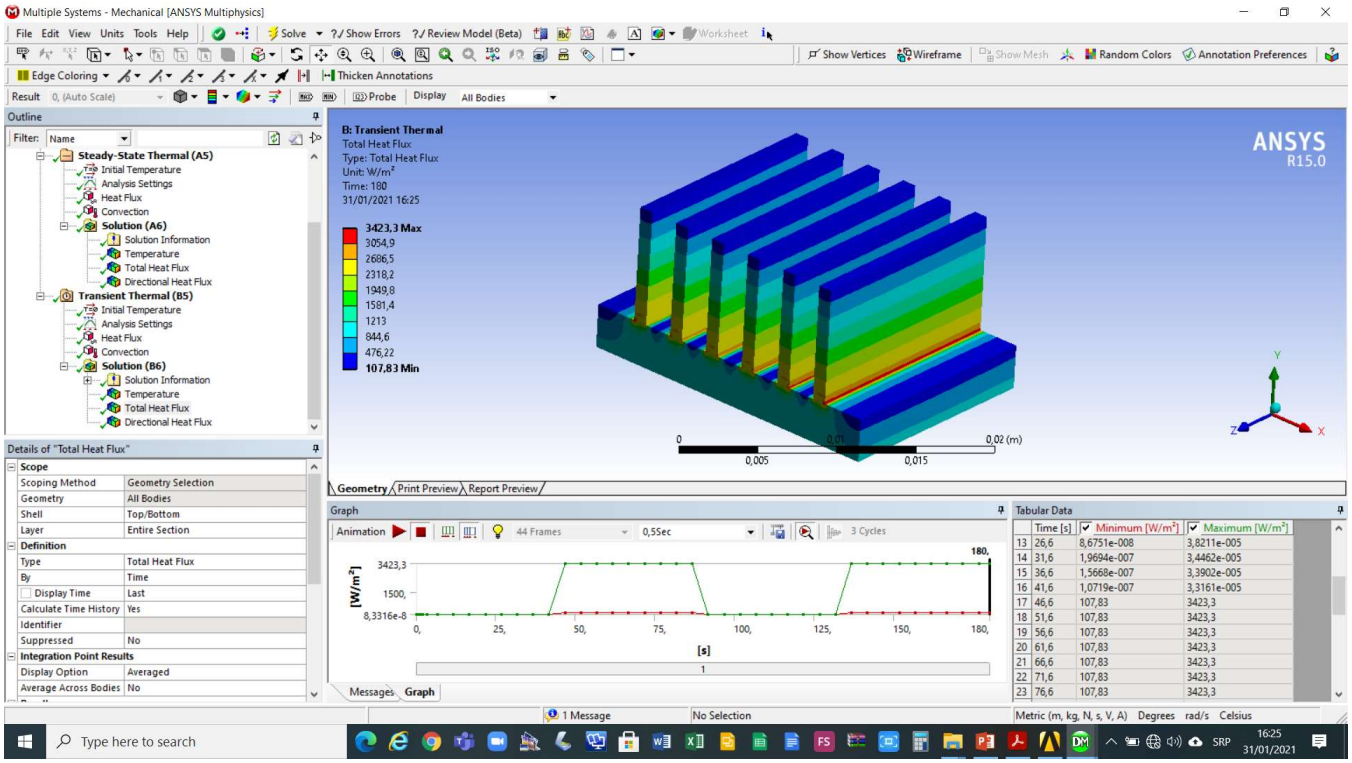
Toplotno opterećenje

Simulacija promjene raspodjele temperature opcija *Graph*



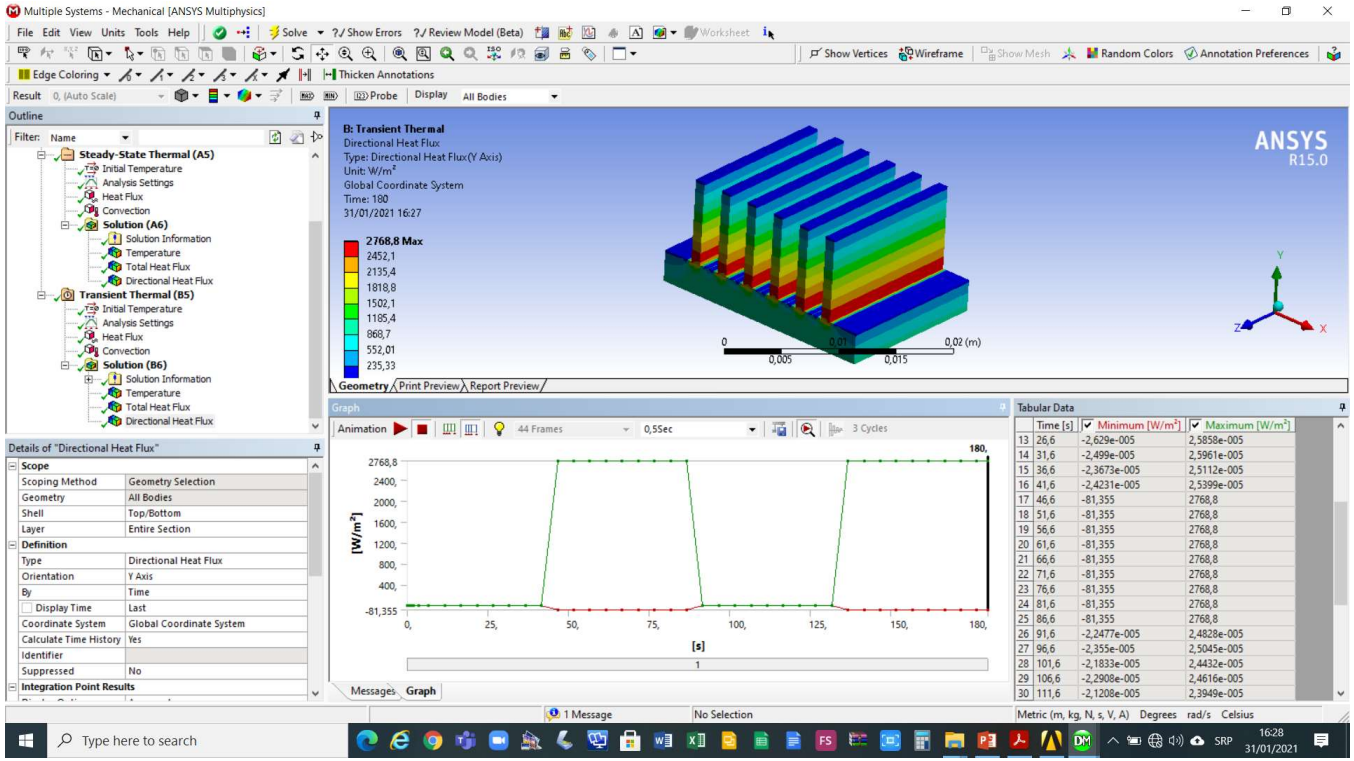
Toplotno opterećenje

Simulacija promjene raspodjele toplotnog fluksa opcija *Graph*



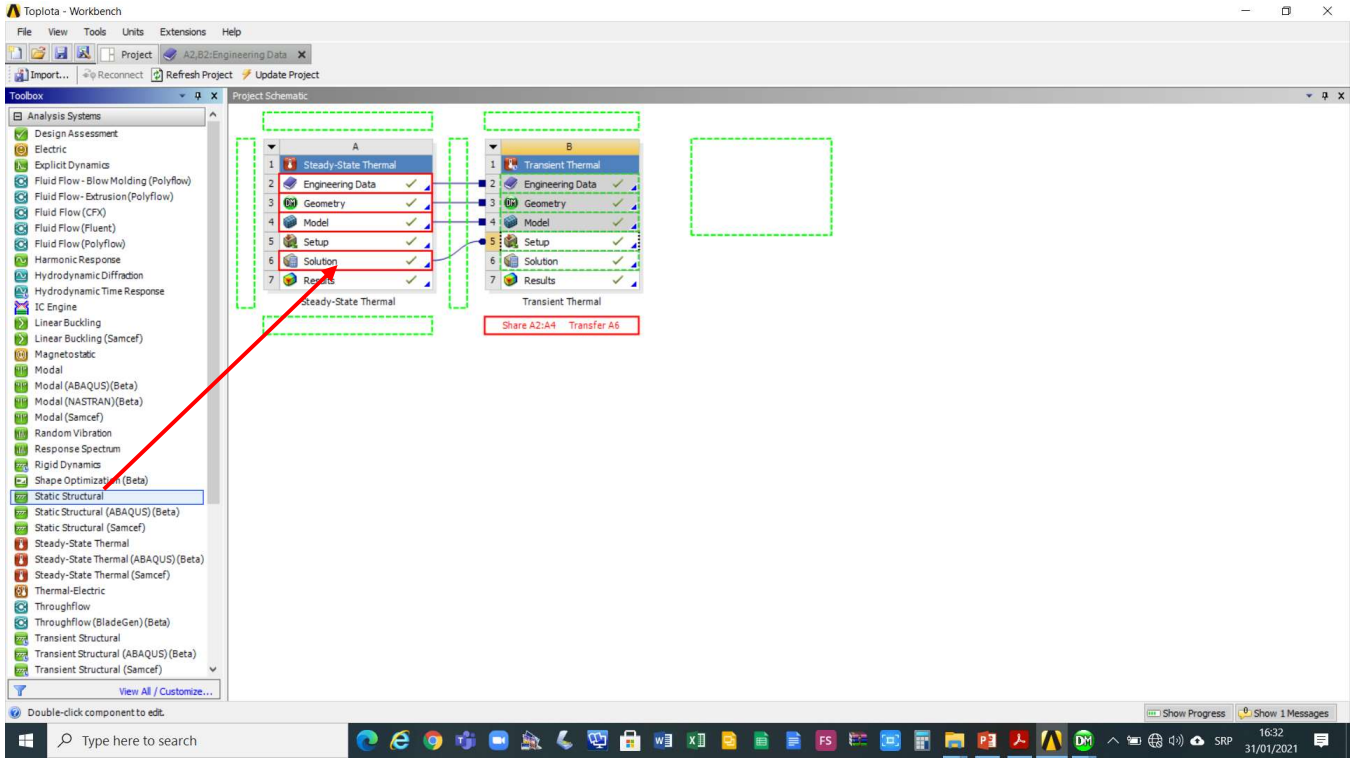
Toplotno opterećenje

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



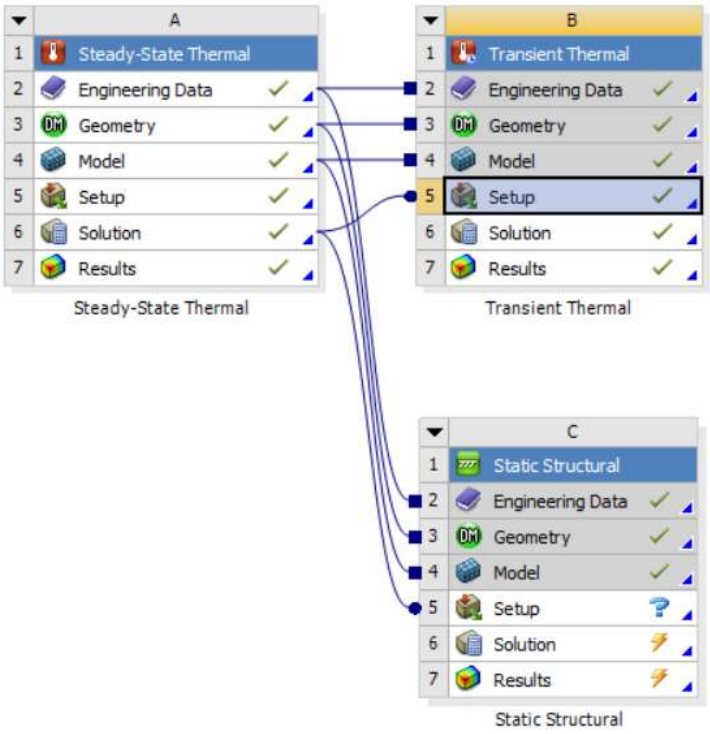
Toplotno opterećenje

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



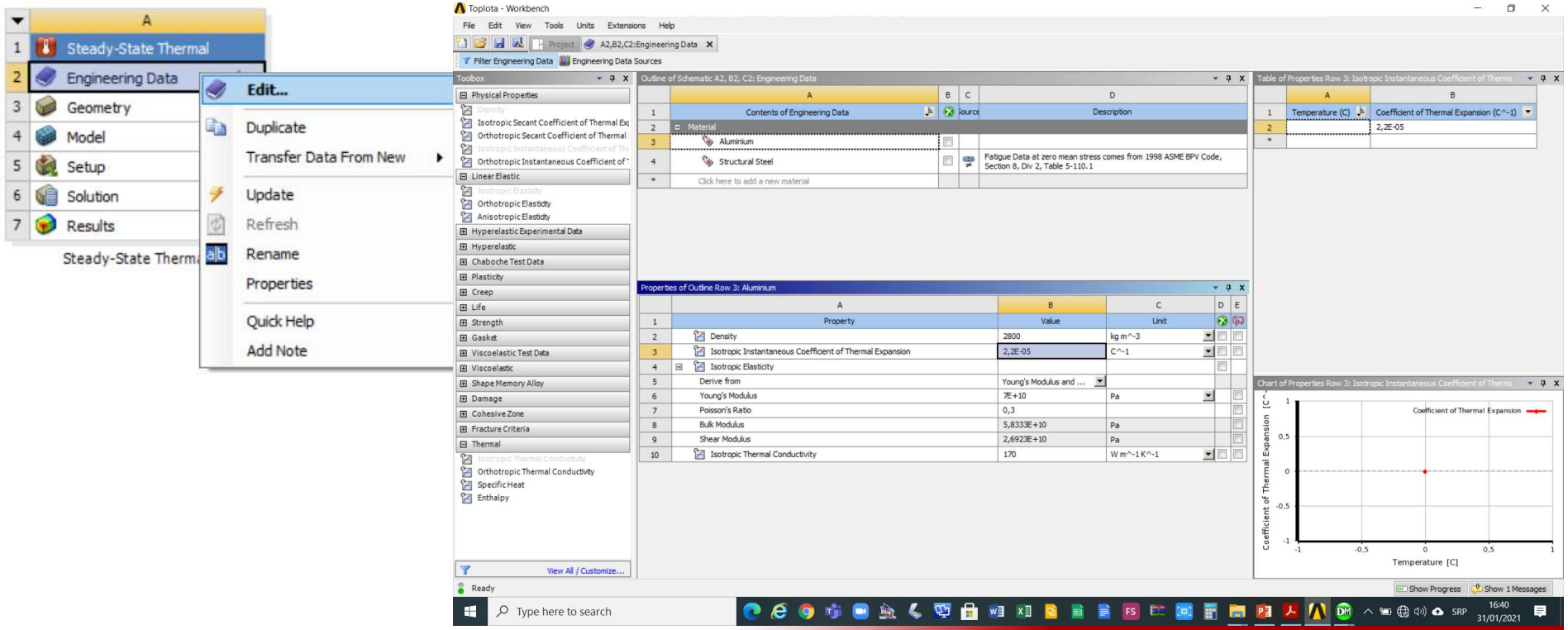
Toplotno opterećenje

Prethodni postupak omogućava potrebno povezivanje stacionarne termalne analize i strukturne analize



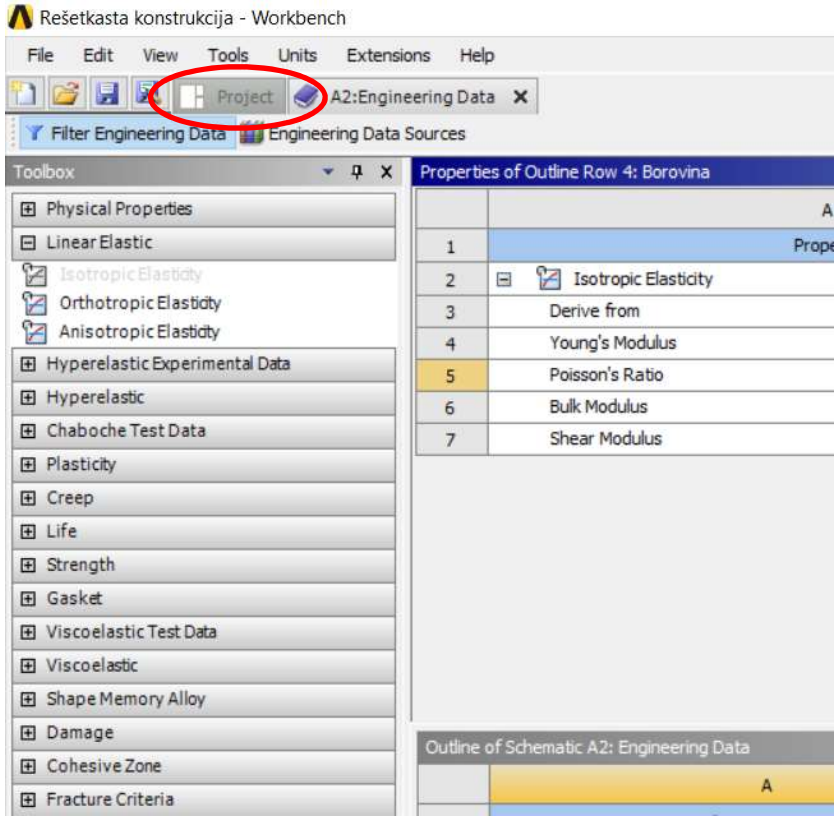
Toplotno opterećenje

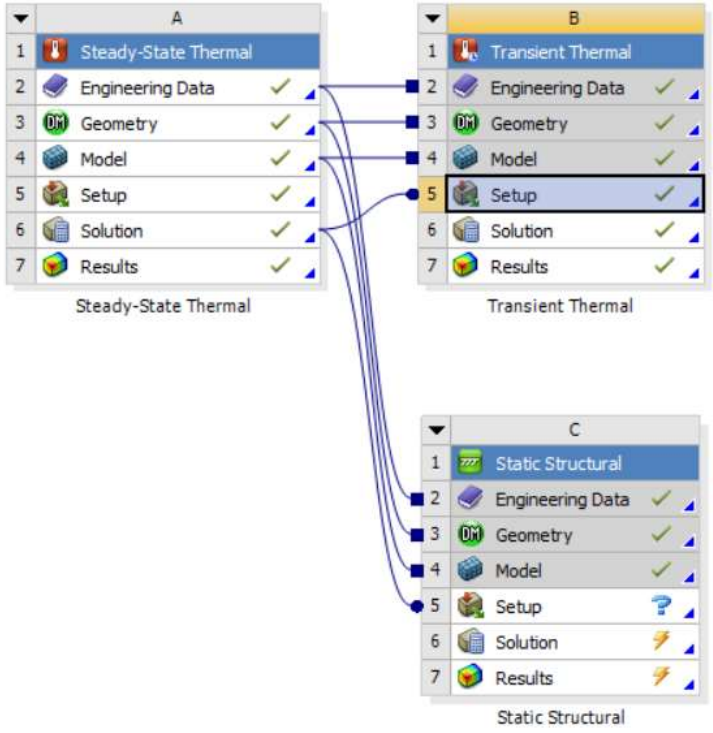
Aktivirati modul Engineering Data i dodati podatke o gustini, modulu elastičnosti, Poisson-ovom koeficijentu, koeficijentu toplotnog širenja



Toplotno opterećenje

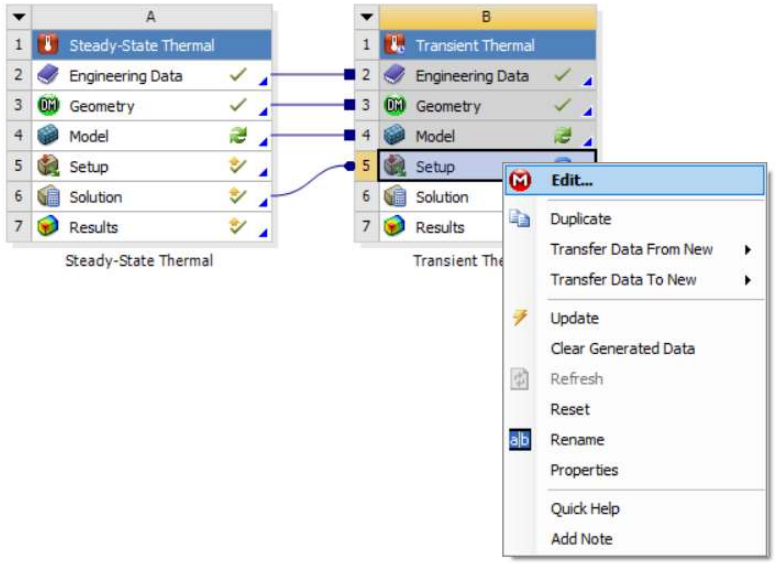
Izabrati opciju *Project* za povratak na shemu projekta

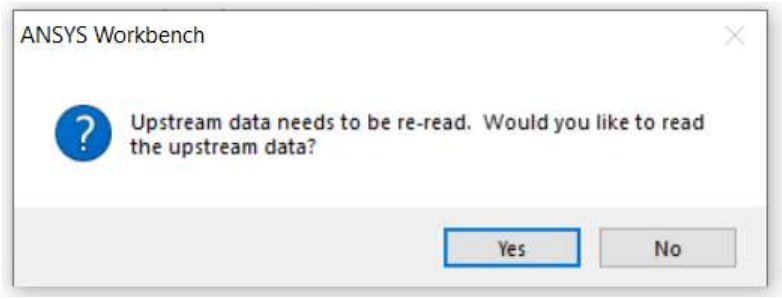




Toplotno opterećenje

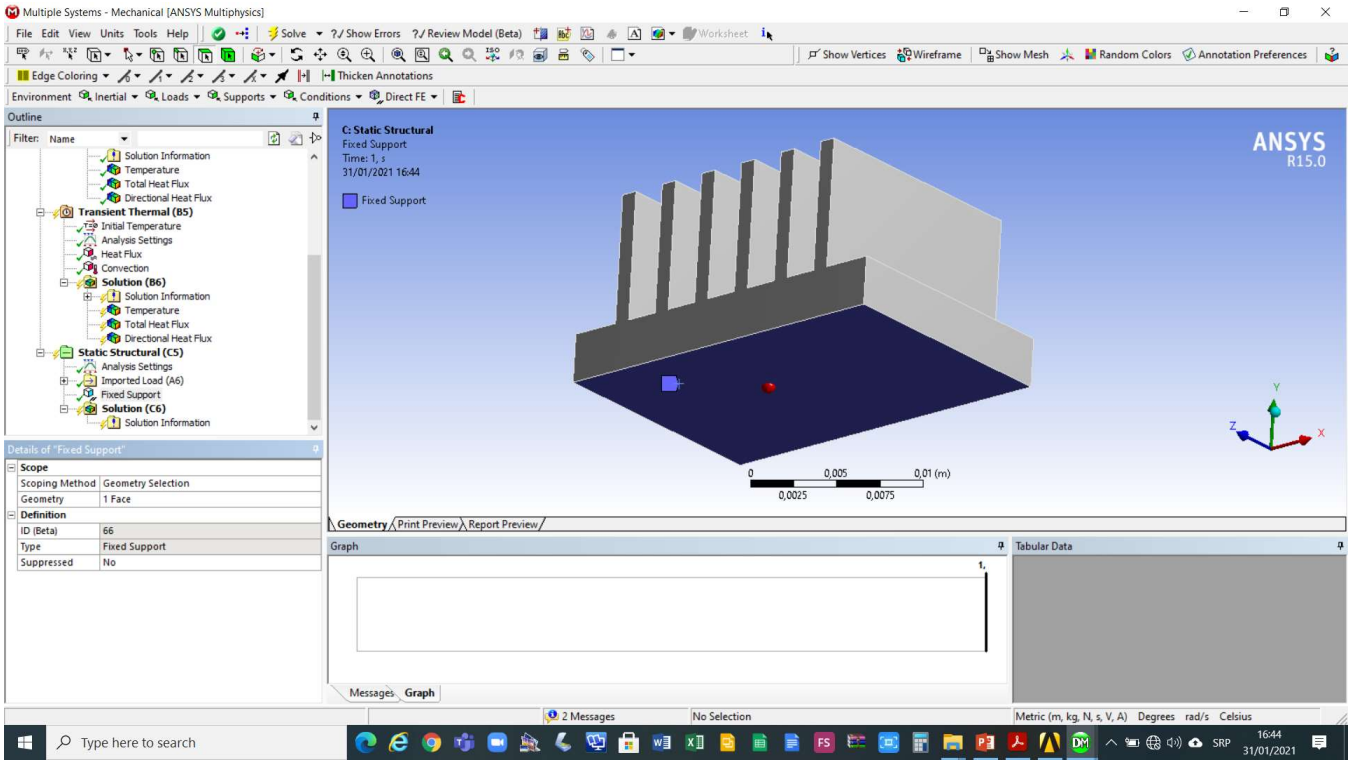
Aktivirati modul Multiple Systems (*Static Structural->Setup->Edit*) i odgovoriti sa *Yes* kako bi se učitali novounešeni podaci





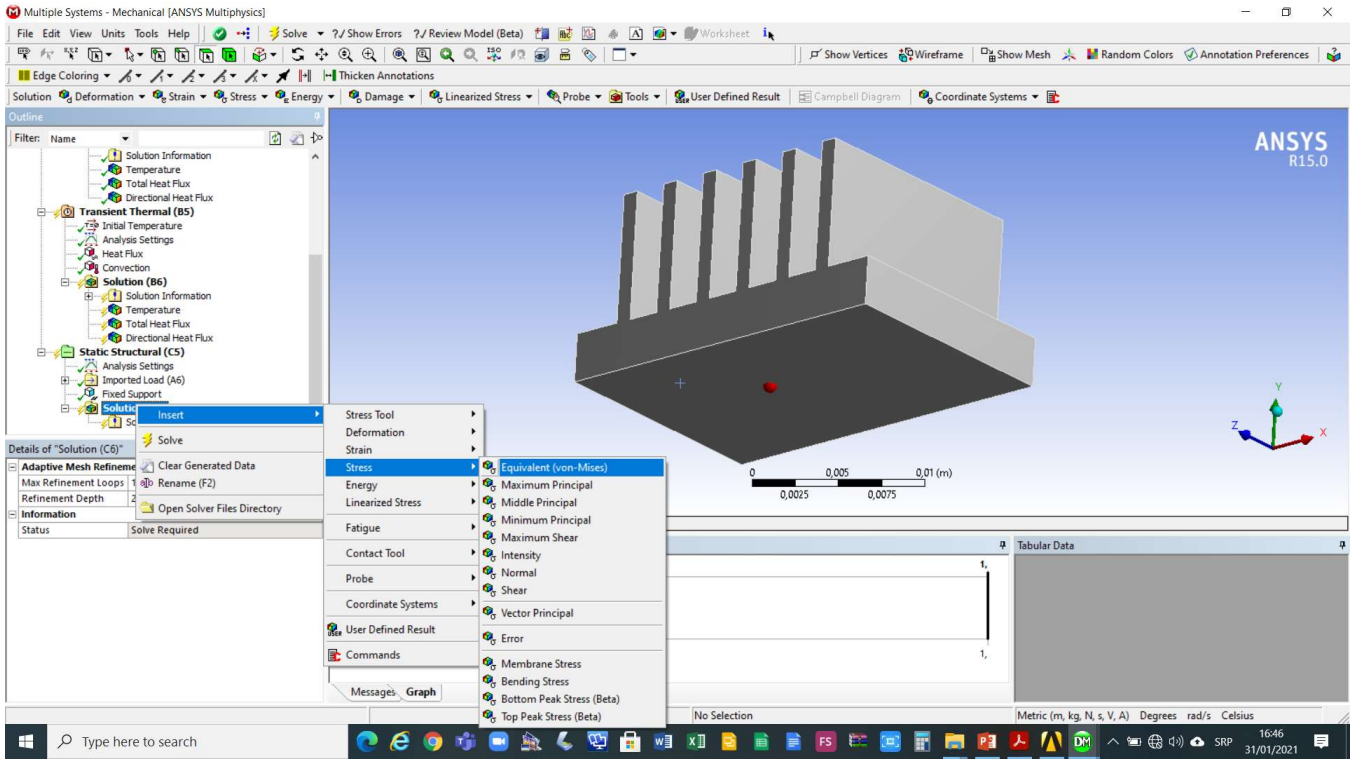
Toplotno opterećenje

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



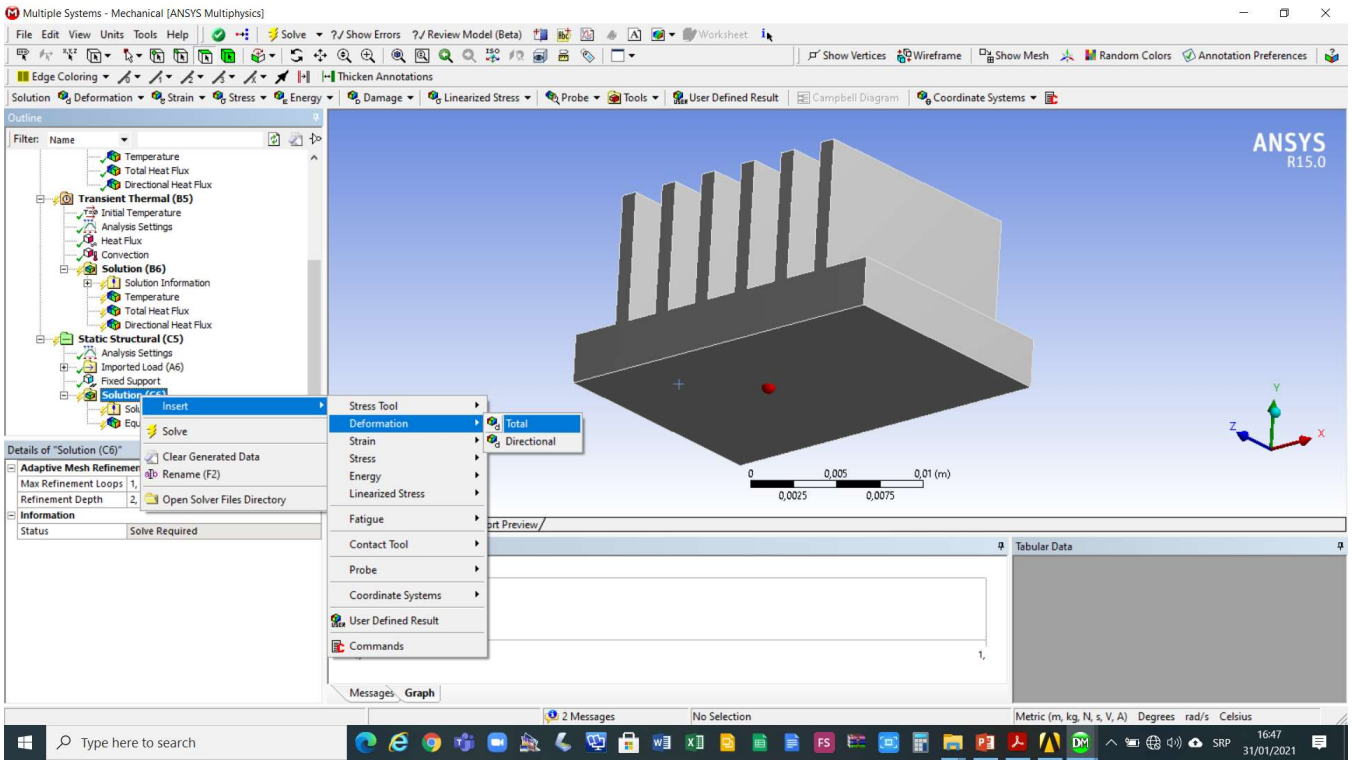
Toplotno opterećenje

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



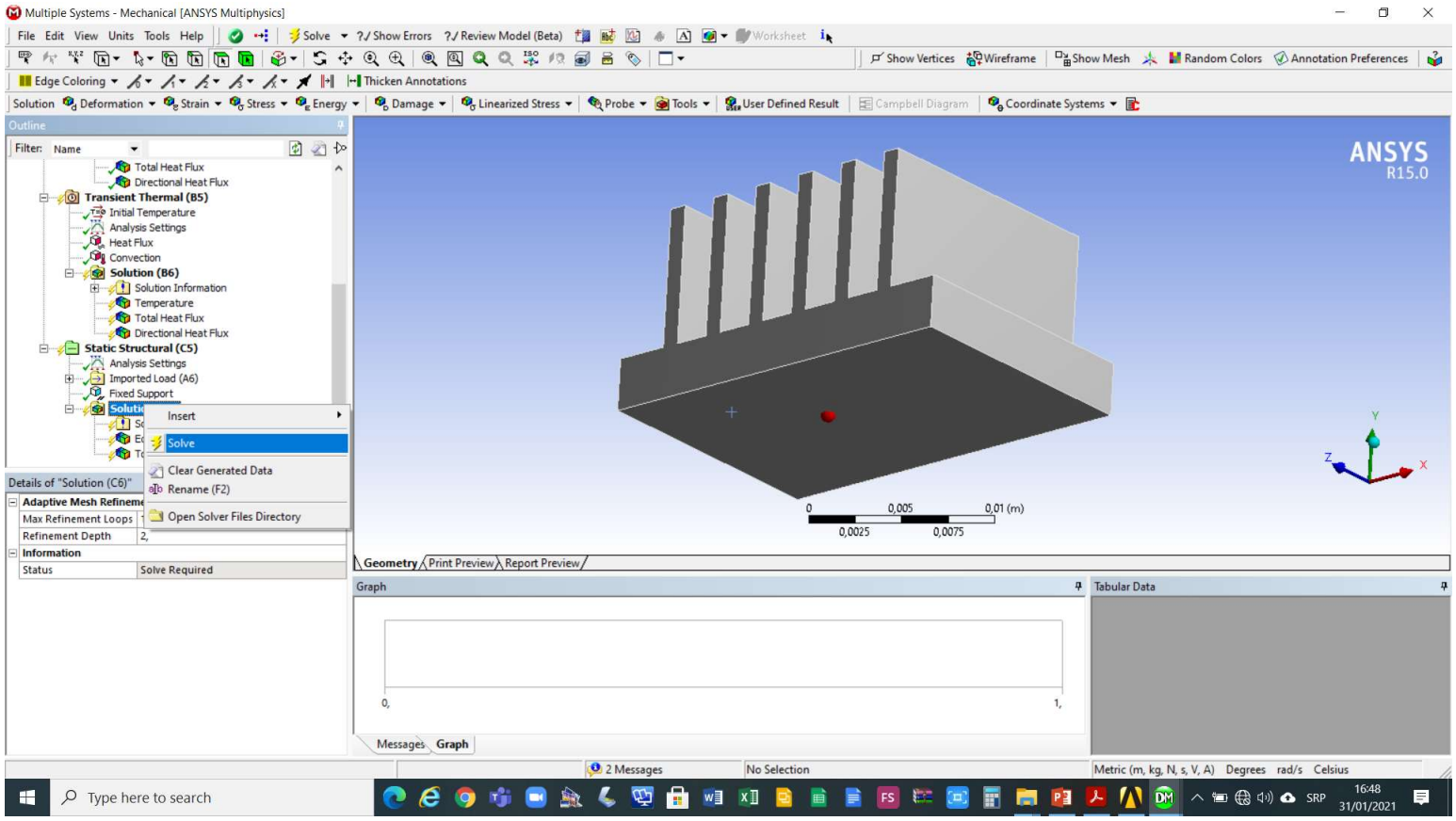
Toplotno opterećenje

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



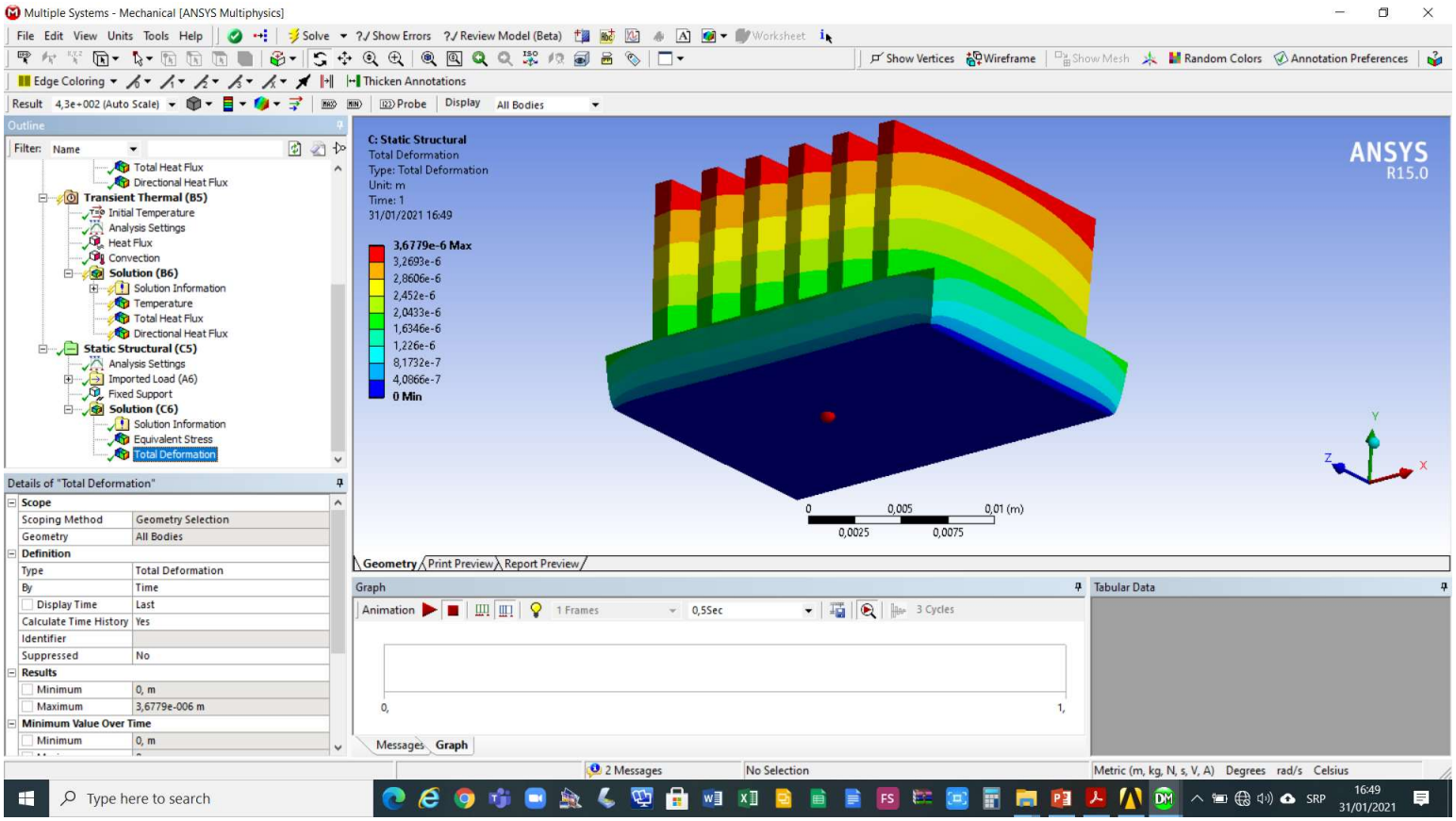
Toplotno opterećenje

Aktivirati izvršenje analize *Solution->Solve*



Toplotno opterećenje

Raspodjela toplotne deformacije



Toplotno opterećenje

Raspodjela toplotnih von Mises-ovih napona