

# Nedjelja 11

## Modeliranje otkaza mašinskog elementa konačnim elementima

# Postavka zadatka

---

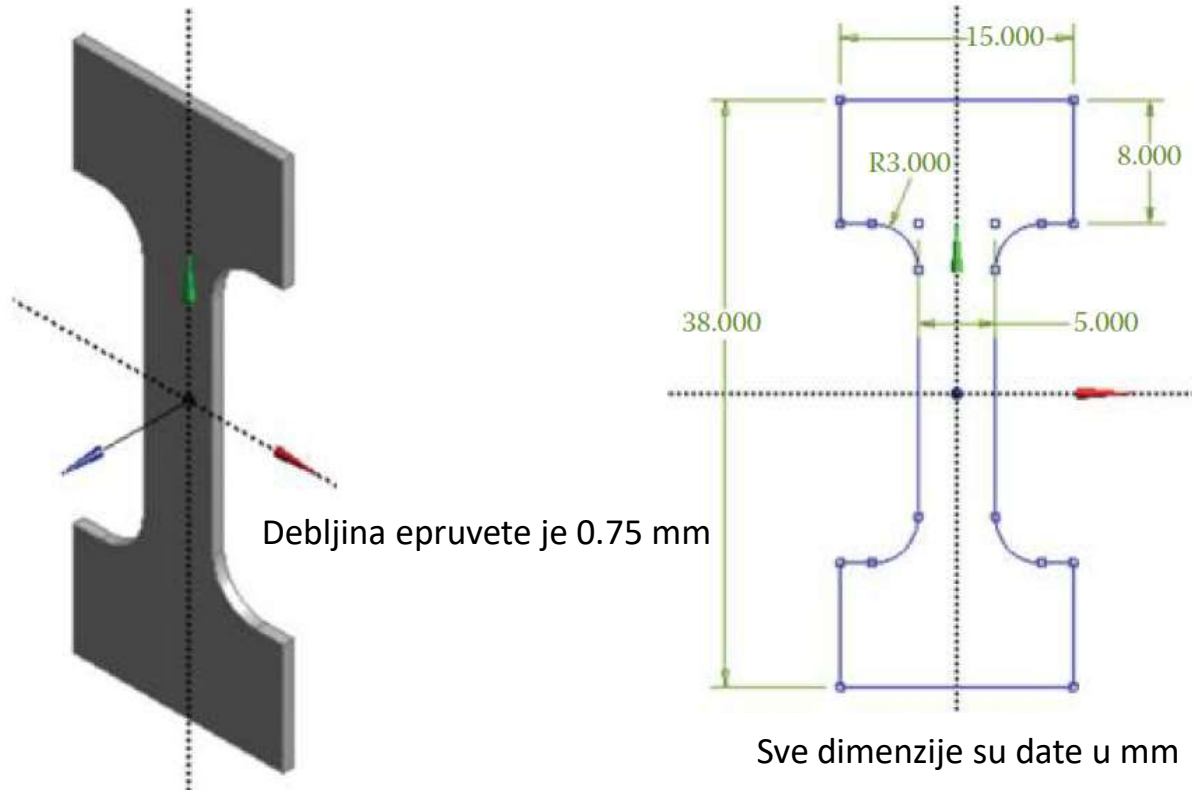
Epruveta je ispitivana na statički, zamorni i otkaz usled izvijanja. Element je izrađen od čelika ( $E=200$  GPa,  $\nu=0.3$ ,  $\rho=7850$  kg/m<sup>3</sup>). Odrediti:

- Da li će epruveta biti plastično deformisana pod dejstvom konstantnog pritiska, da li će doći do njenog izvijanja i naći prva tri oblika izvijanja
- Dužinu radnog vijeka ukoliko je pritisak naizmjenično promjenljiv i da li će doći do zamornog otkaza ukoliko je projektovani radni vijek  $10^6$  ciklusa

# Postavka zadatka

**Granični uslovi:** Nepokretni oslonci po donjoj površini

**Opterećenje:** Pritisak od 50 MPa djeluje po gornjoj površini



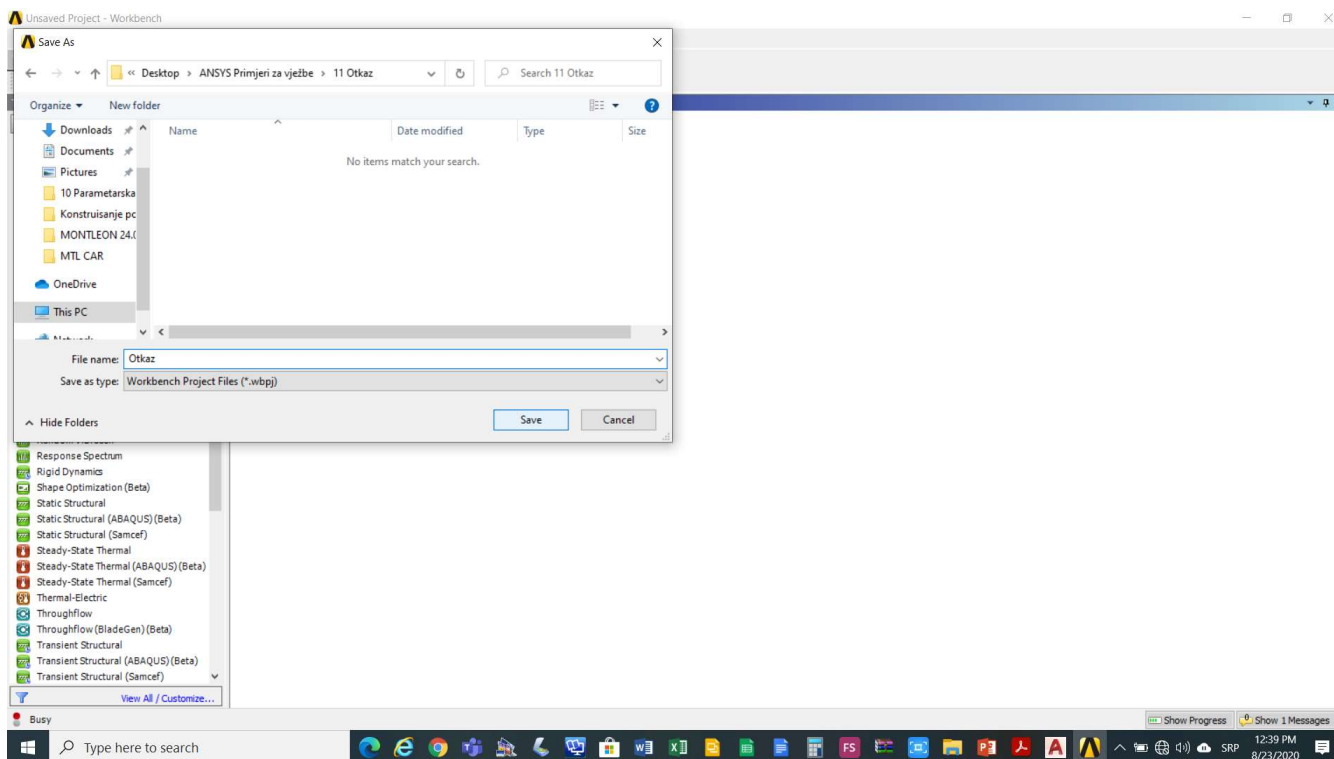
# Postavka zadatka

---



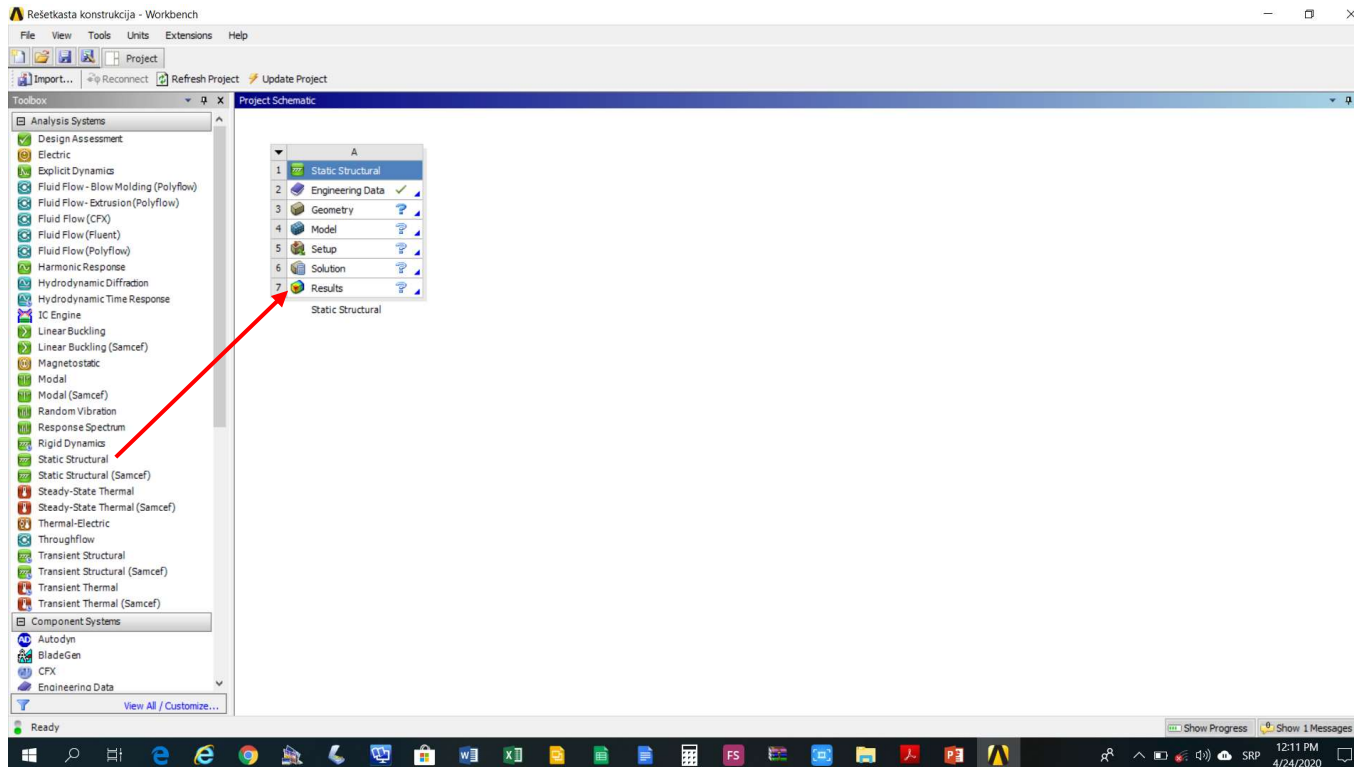
# Statički otkaz

Aktivirati program ANSYS i sačuvati prazan projekat pod nazivom Otkaz



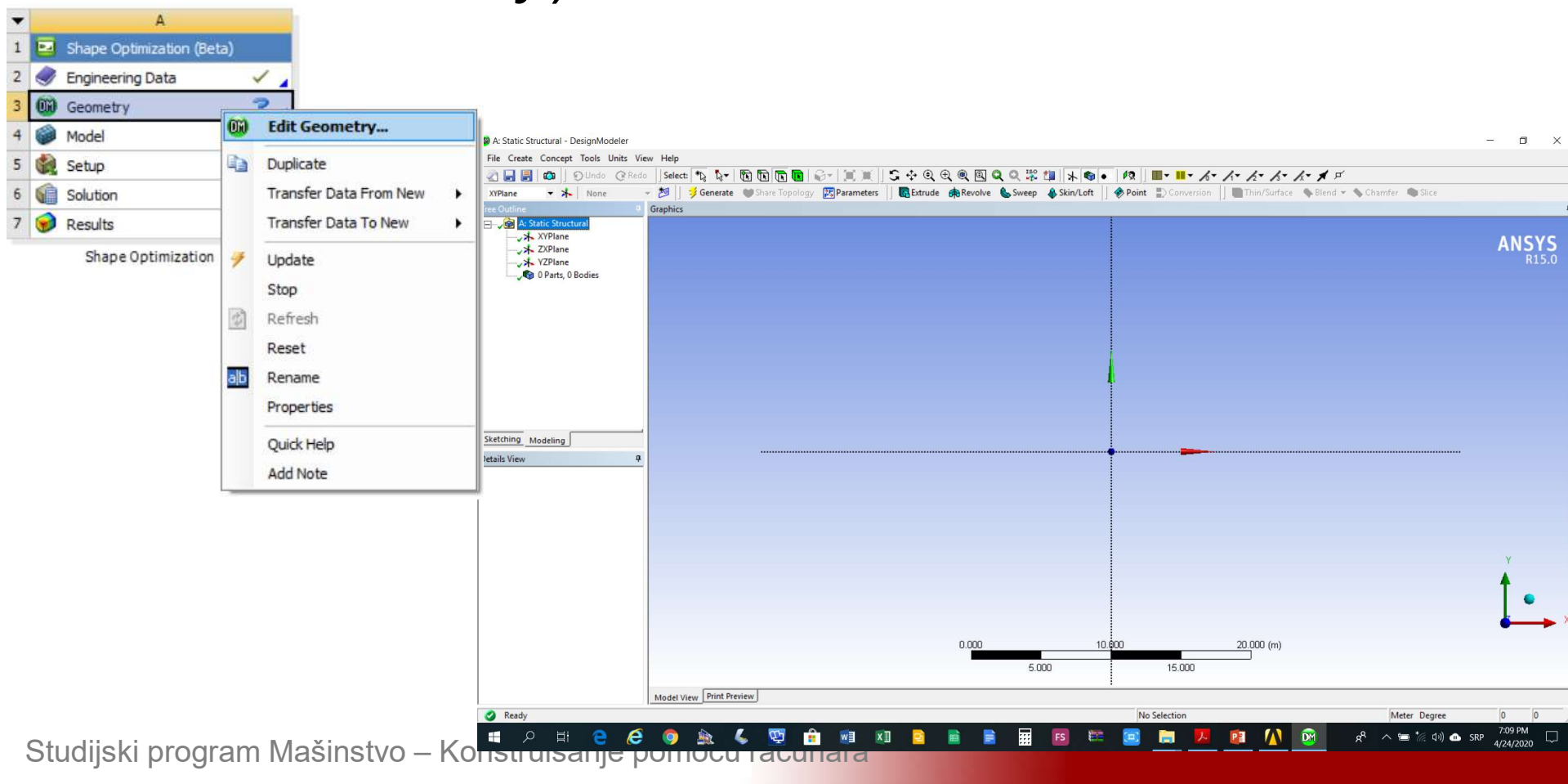
# Statički otkaz

Kreirati statičku linearnu analizu (*Static Structural*) na shemi projekta (*Project Schematic*)



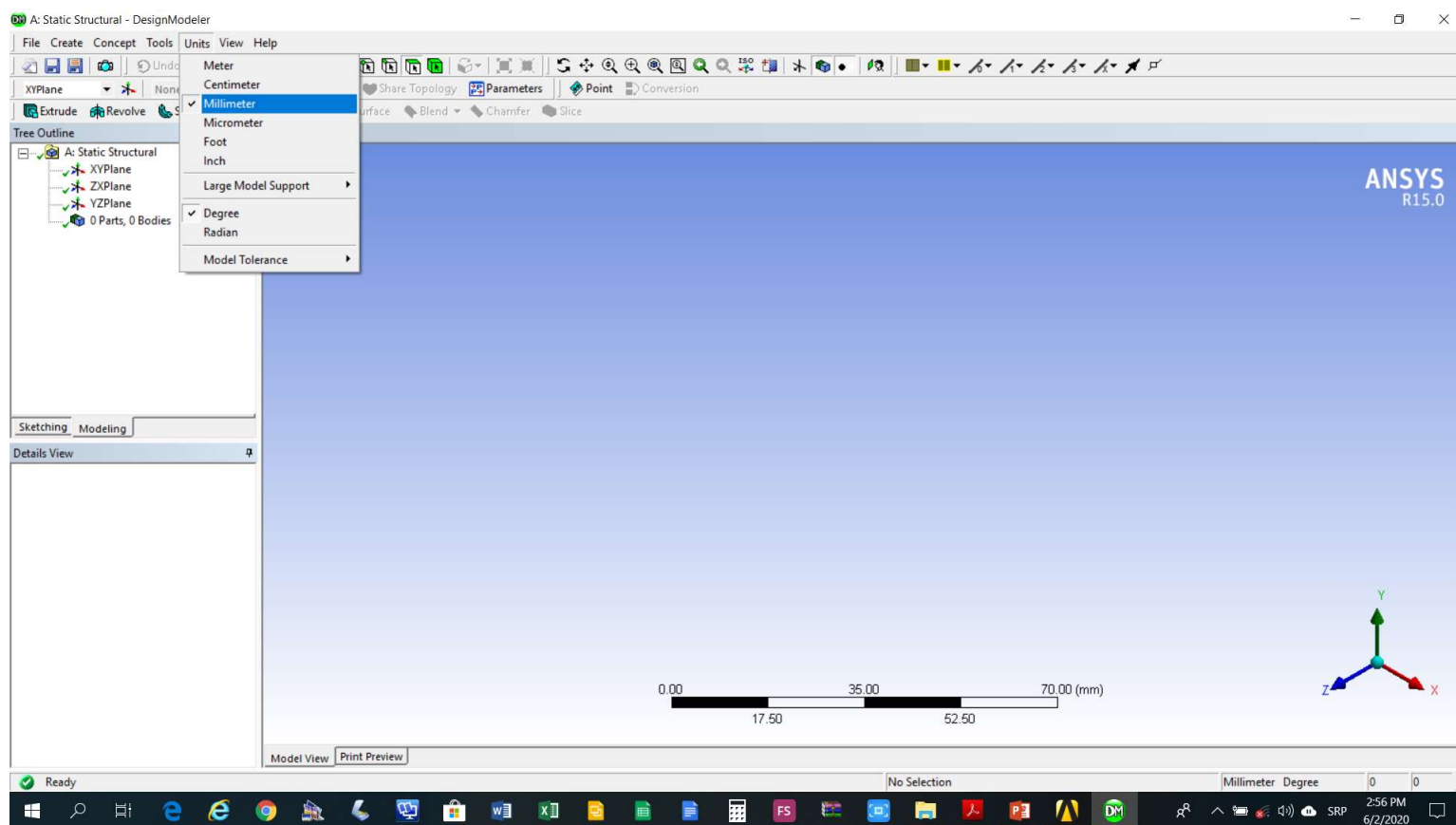
# Statički otkaz

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



# Statički otkaz

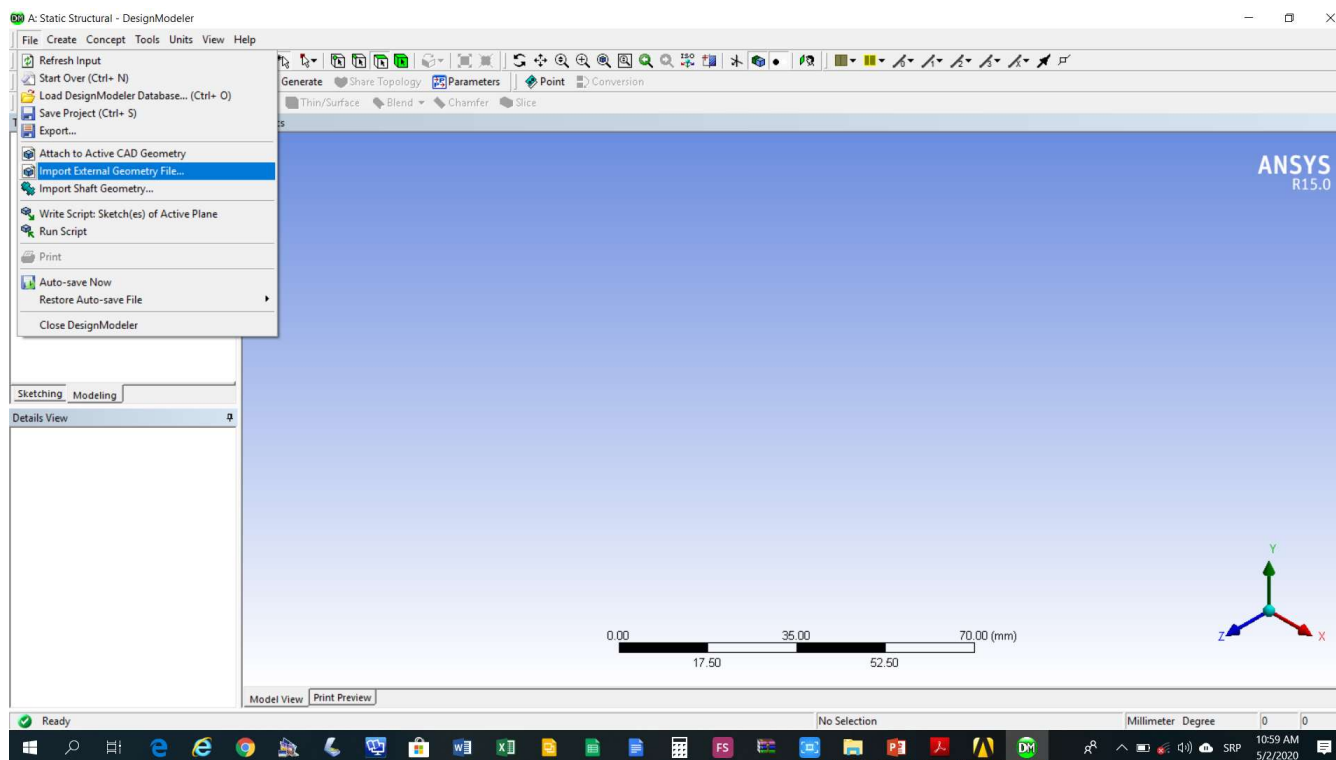
Podesiti dužinske jedinice (Units->*Milimeter*)





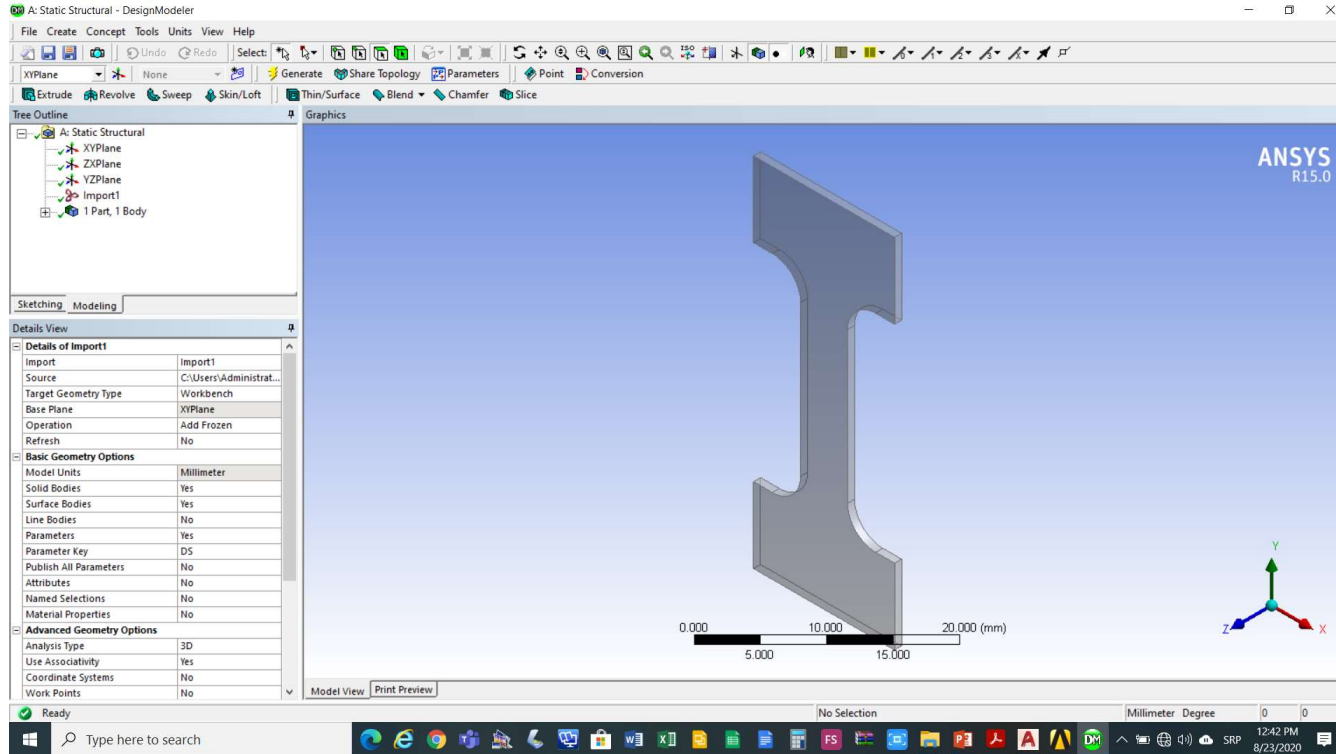
# Statički otkaz

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



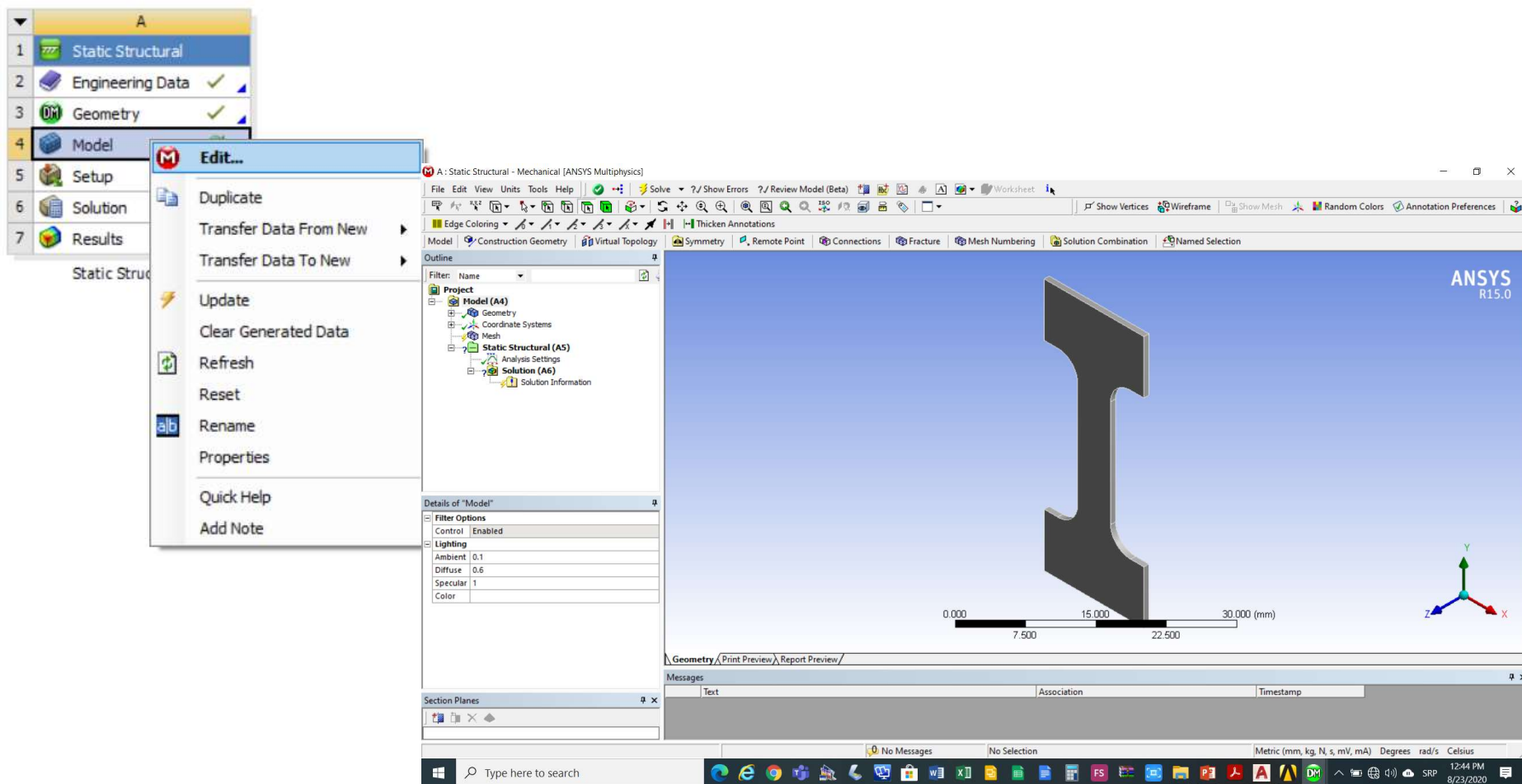
# Statički otkaz

Učitavanje eksterno generisane geometrije  
okončati komandom *Generate*



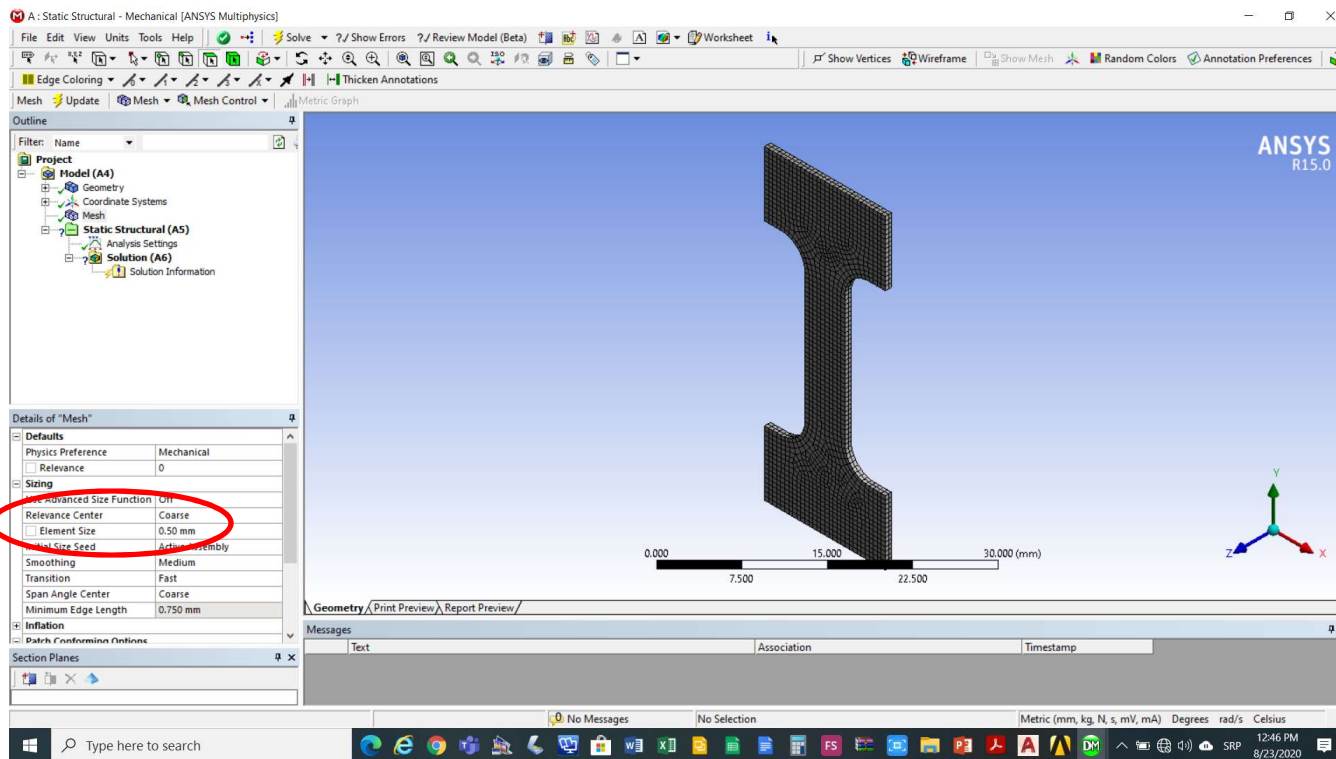
# Statički otkaz

Aktivirati modul Static Structural (*Model*->*Edit*)



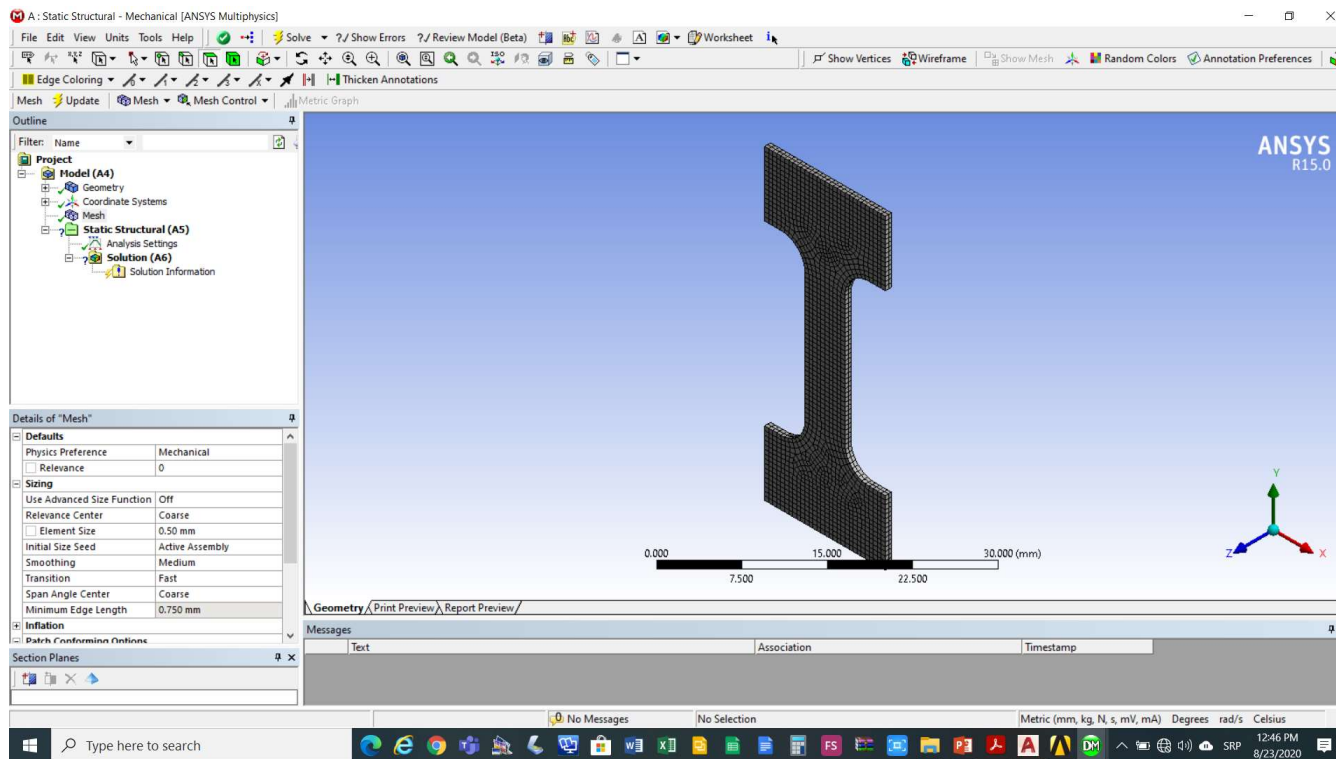
# Statički otkaz

Zadati veličinu konačnih elemenata *Details of Mesh->Sizing->Element Size* na 0.5 mm



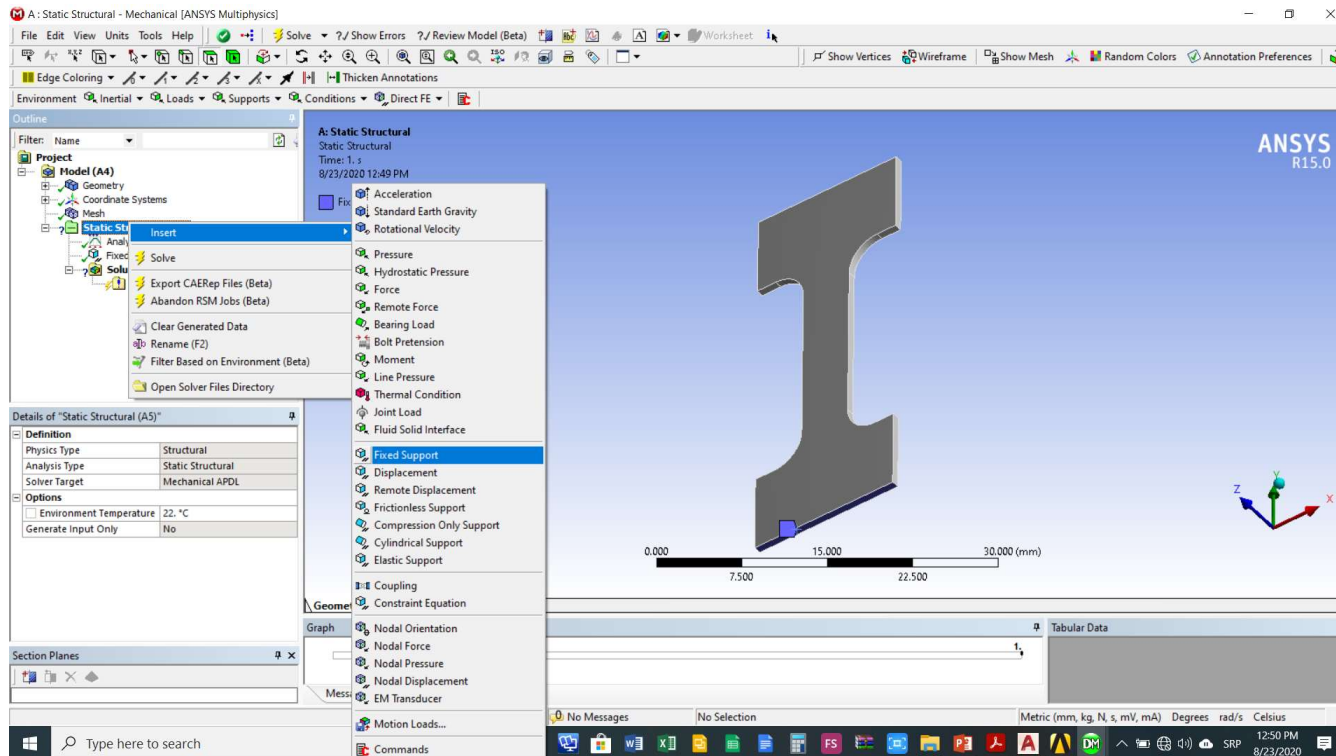
# Statički otkaz

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



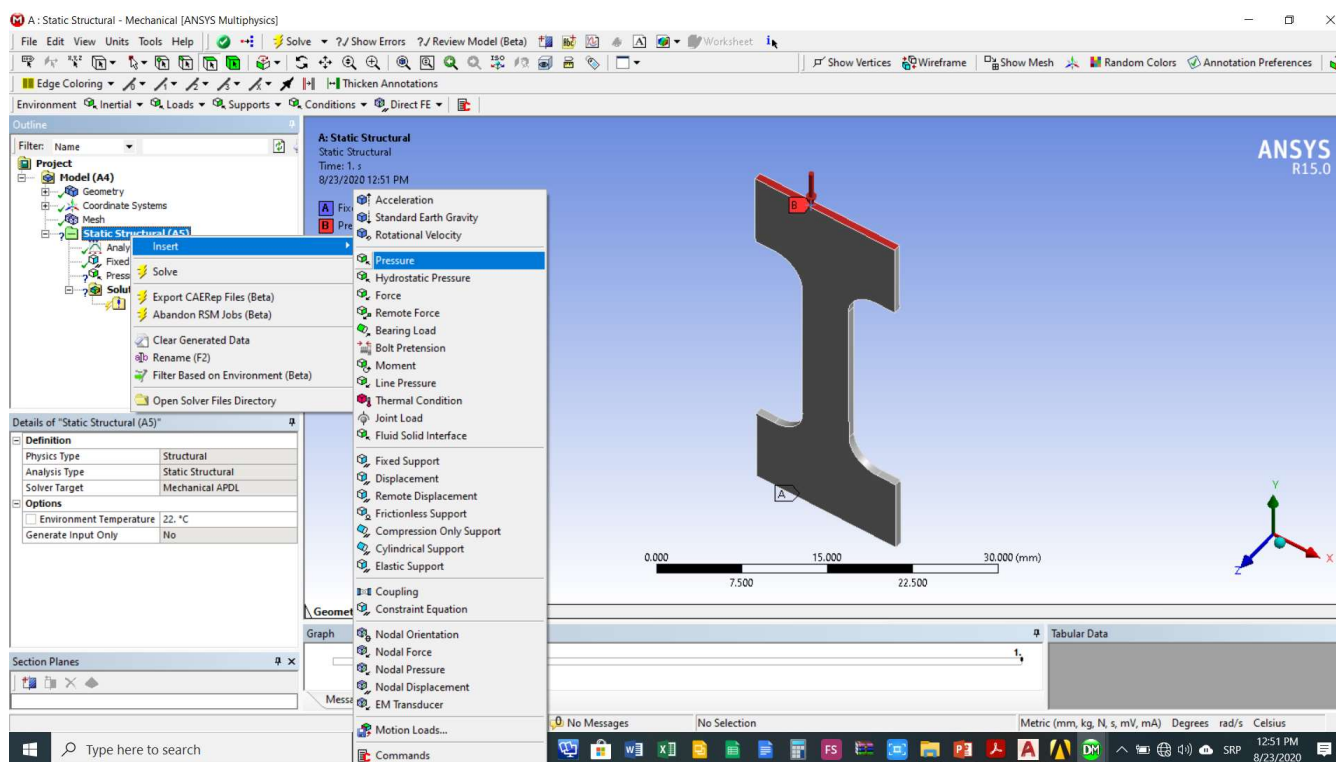
# Statički otkaz

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



# Statički otkaz

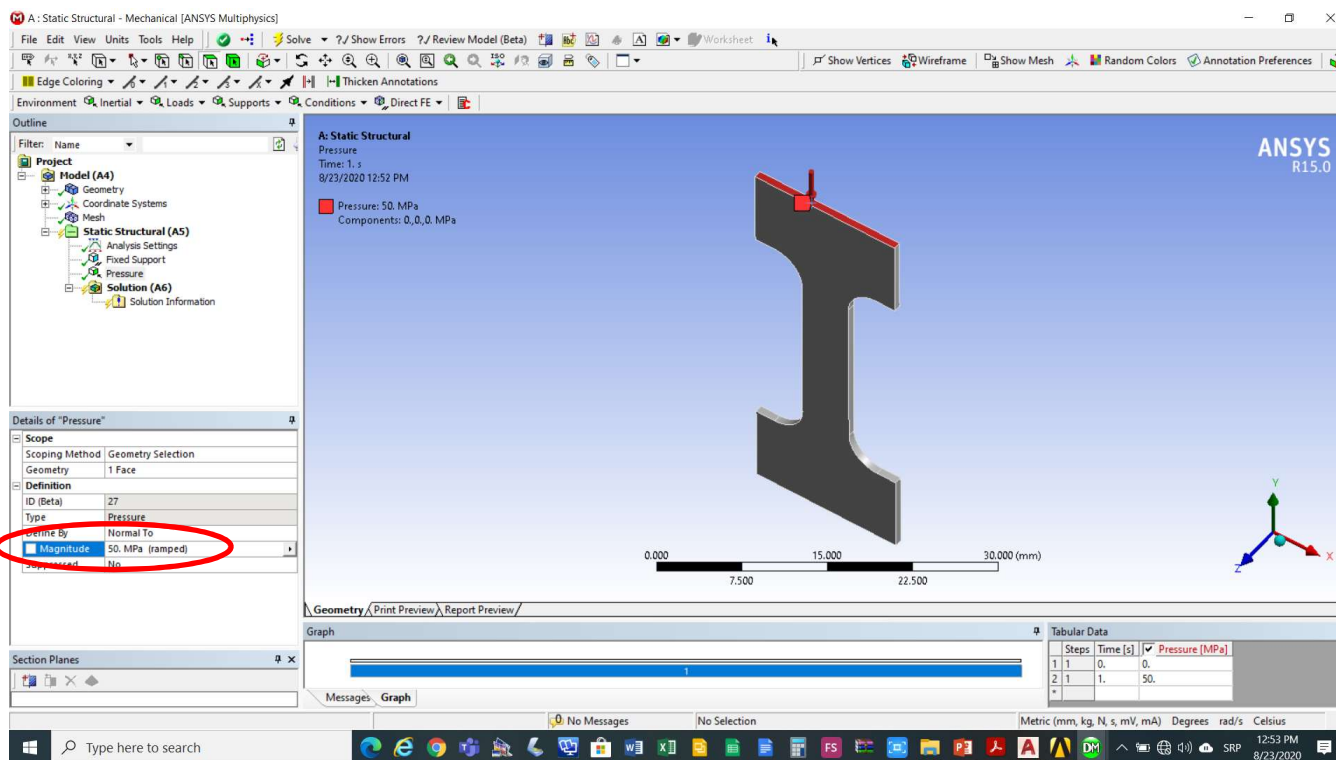
Dodati silu na krajnju gornju površinu *Static Structural->Insert->Pressure*





# Statički otkaz

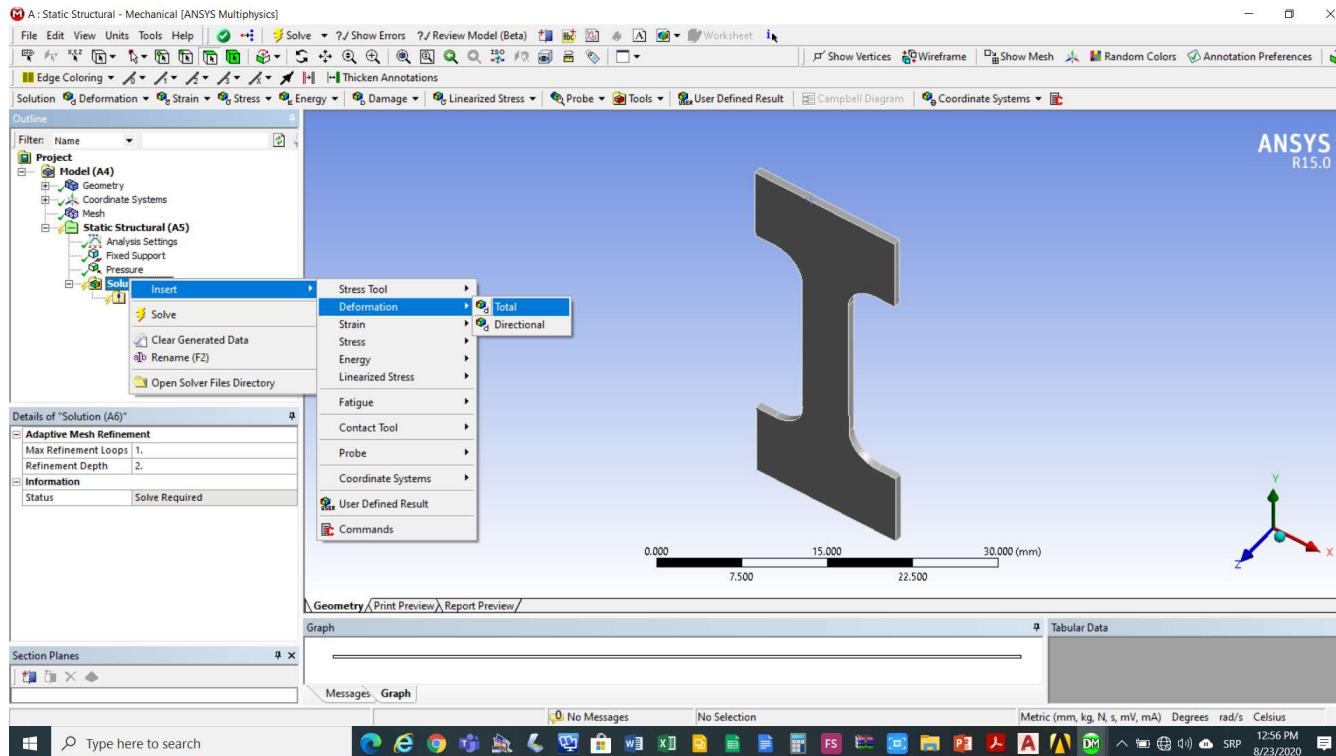
Podesiti intezitet sile *Details of Pressure->Definition->Magnitude* na 50 MPa





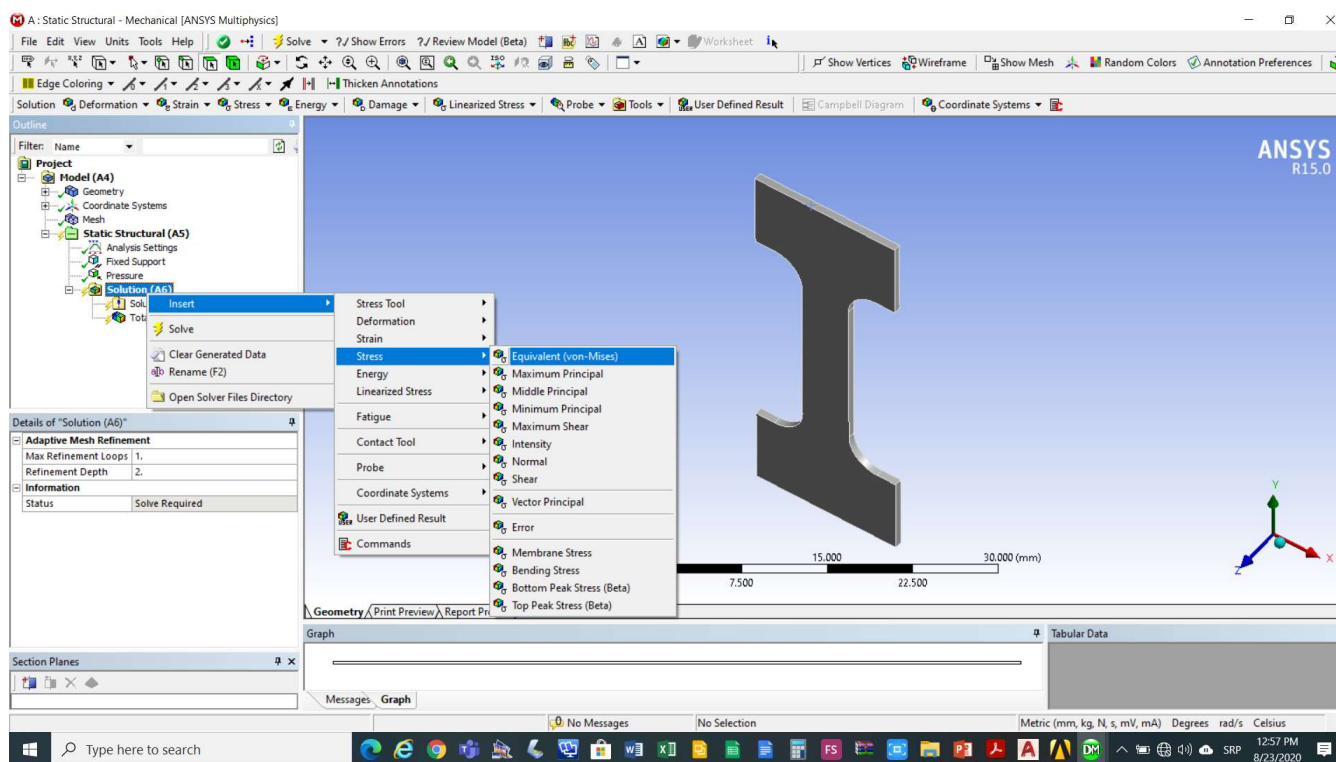
# Statički otkaz

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



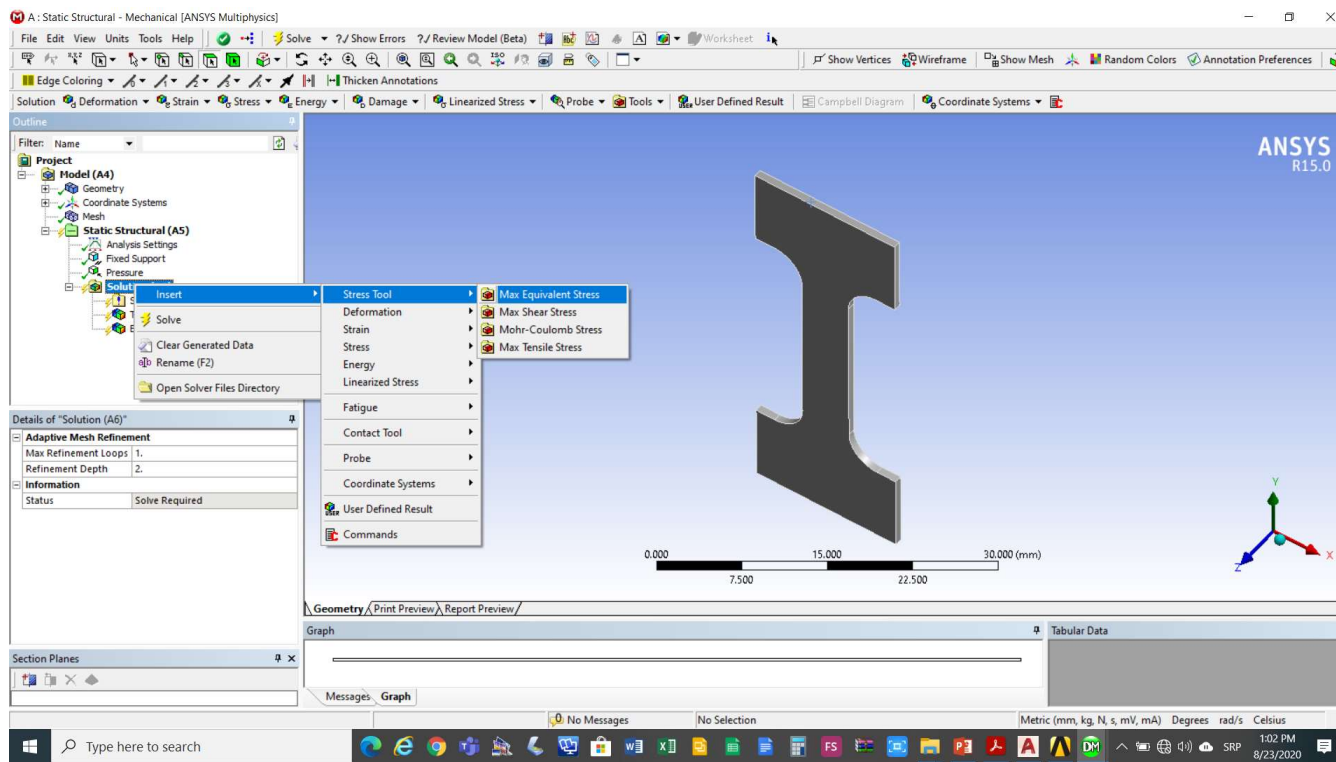
# Statički otkaz

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



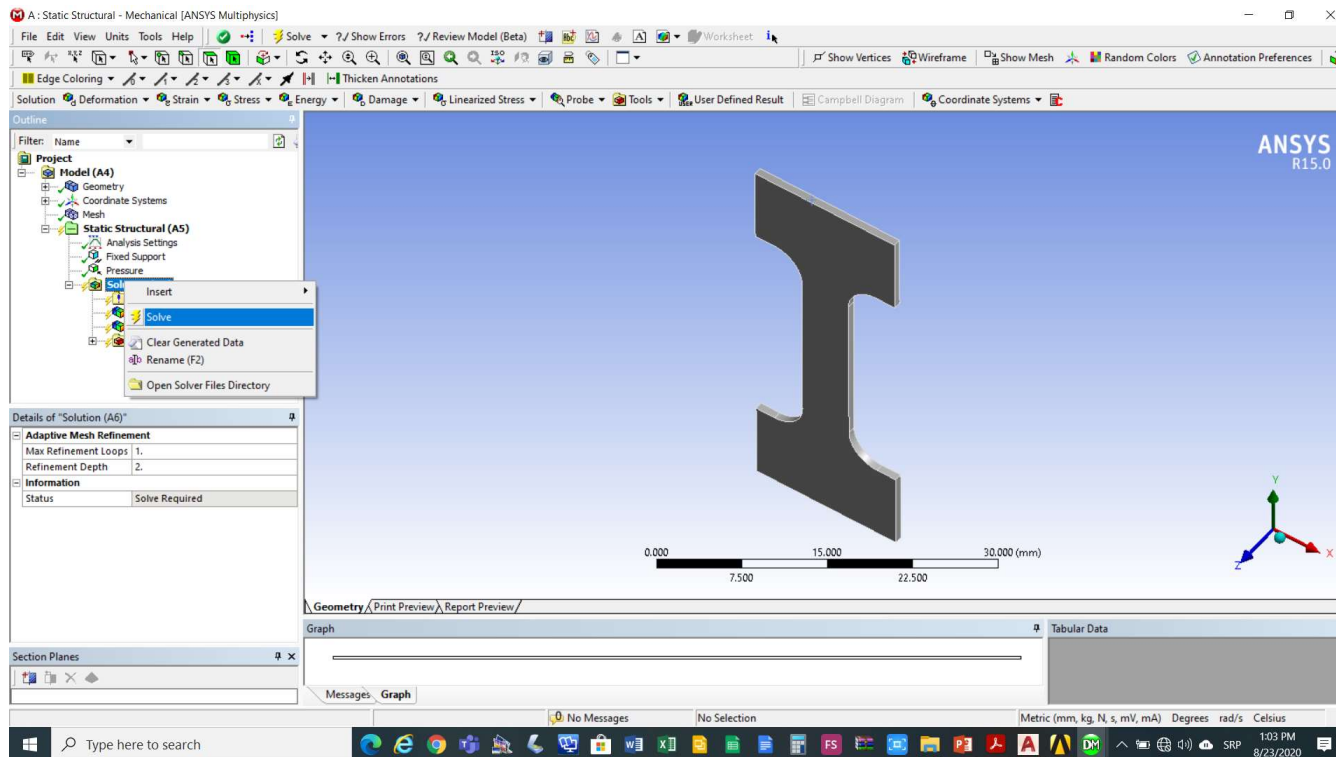
# Statički otkaz

Dodati određivanje stepena sigurnosti u odnosu na napon na granicu tečenja *Solution->Insert->Stress Tool->Max Equivalent Stress*



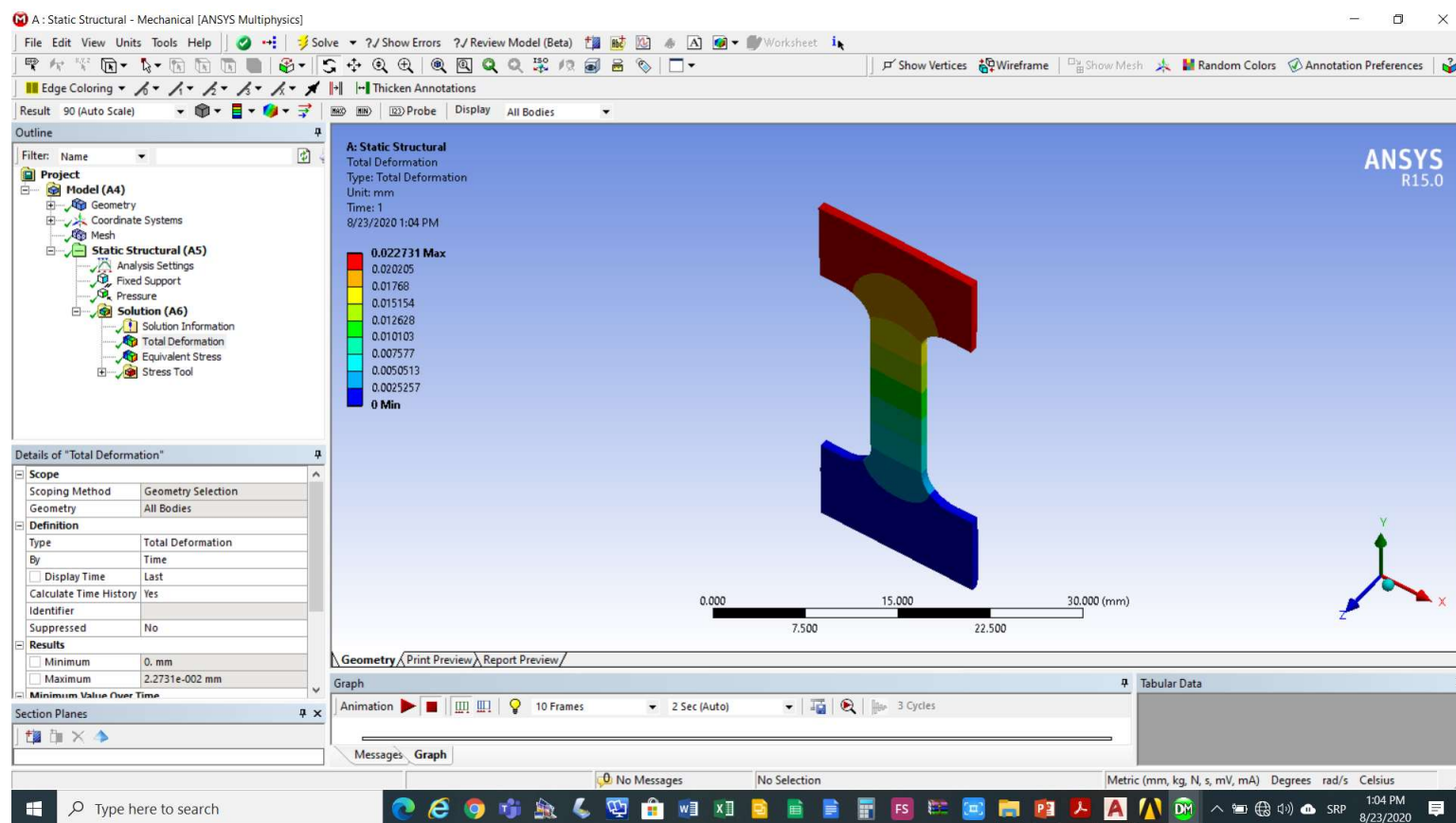
# Statički otkaz

Aktivirati izvršenje analize *Solution*->*Solve*



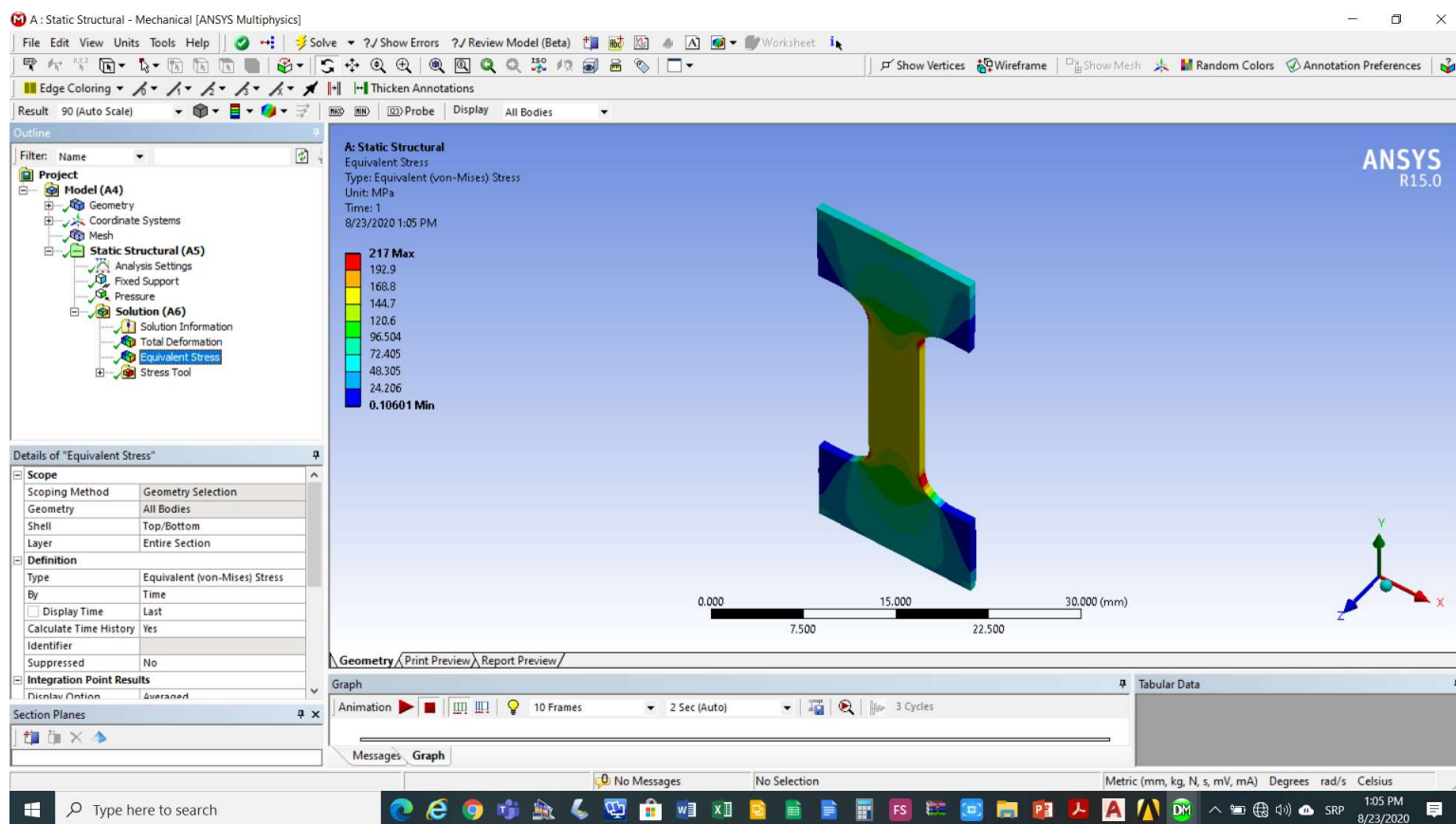
# Statički otkaz

## Raspodjela pomjeranja



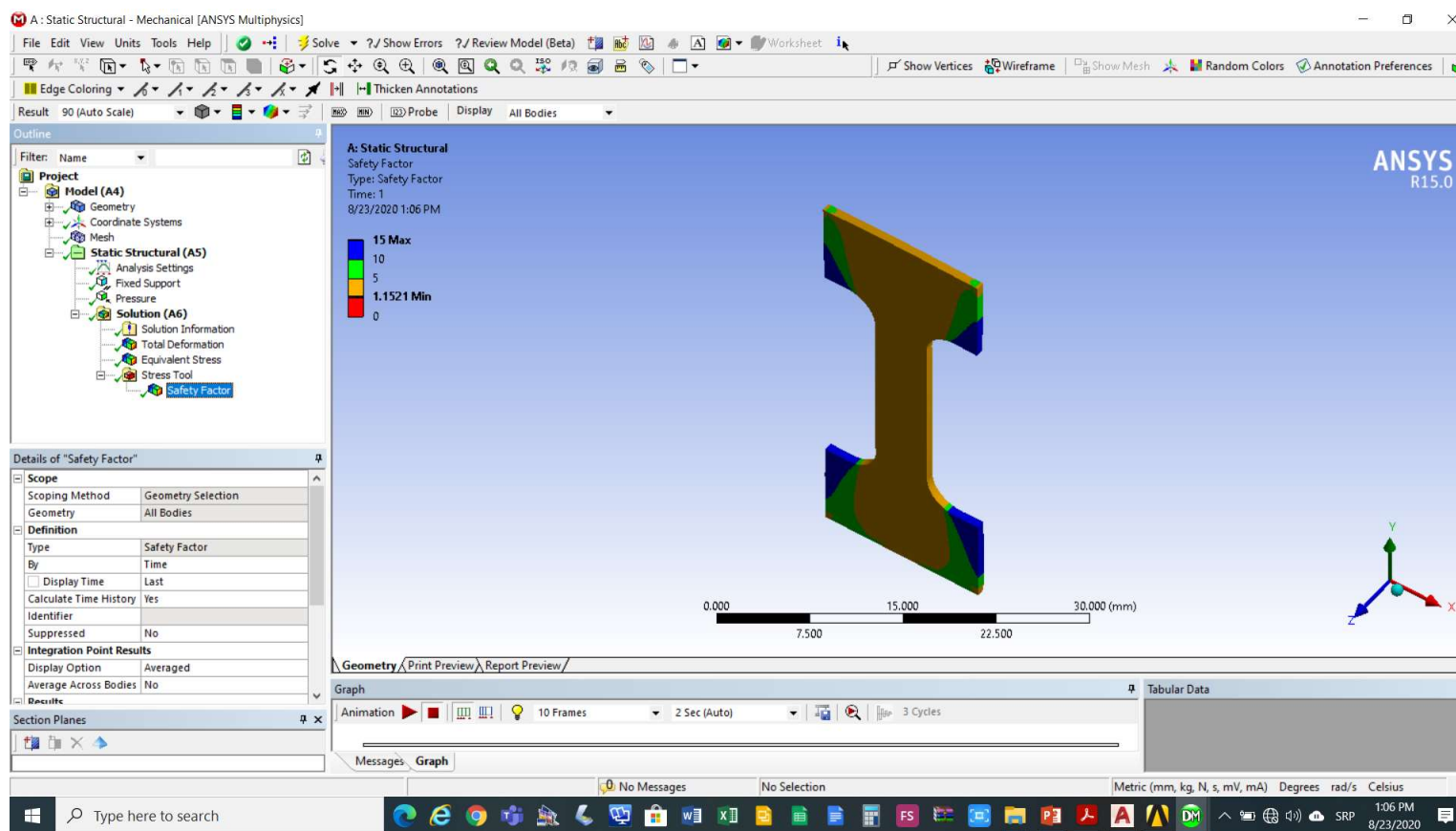
# Statički otkaz

## Raspodjela *von Mises*-ovih napona



# Statički otkaz

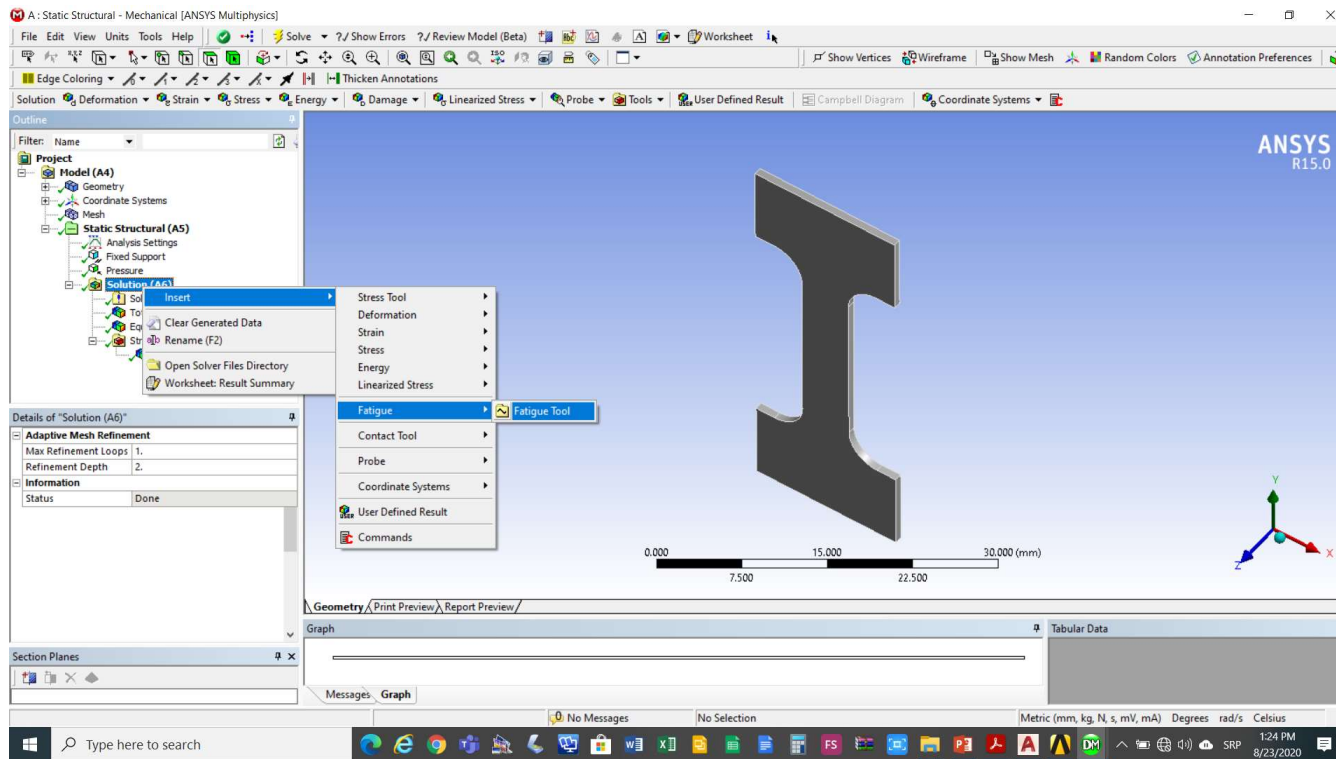
Raspodjela stepena sigurnosti u odnosu na plastični deformaciju





# Zamorni otkaz

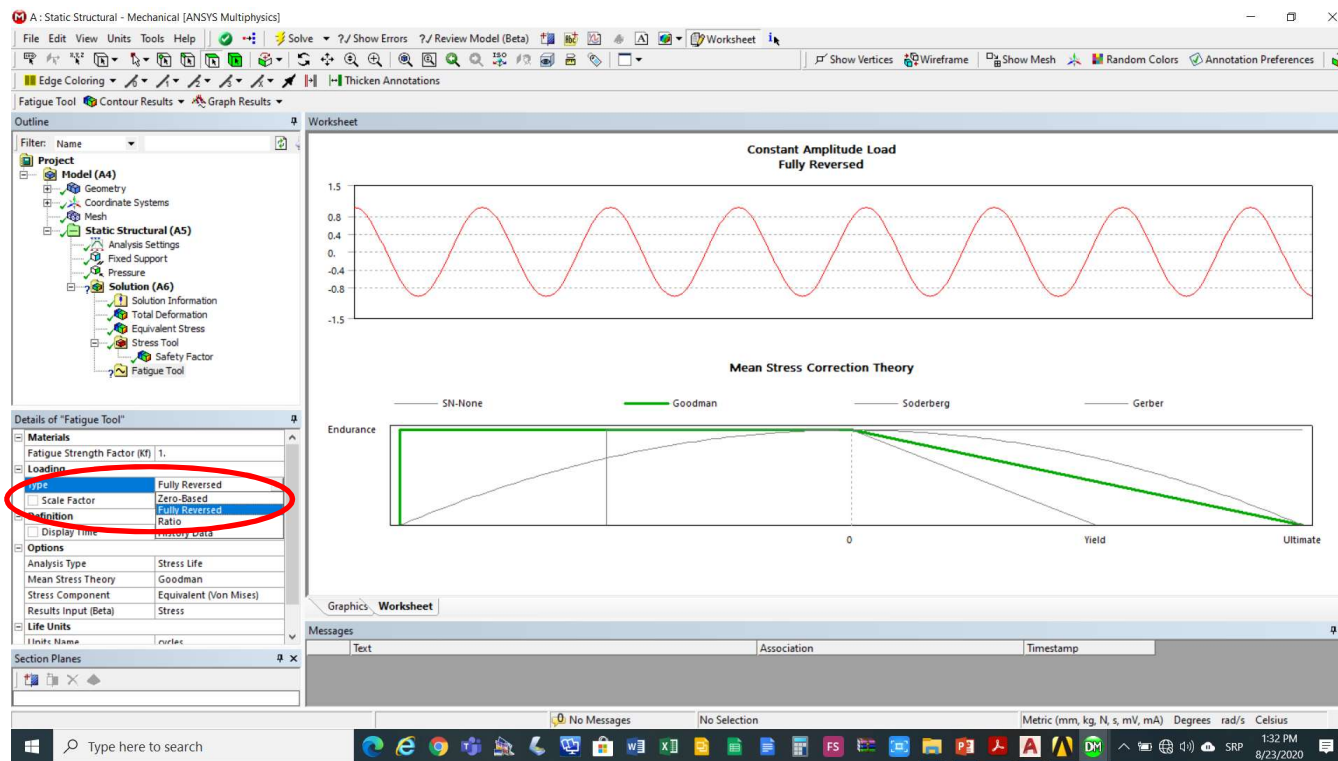
Dodati analizu zamornog otkaza *Solution->Insert->Fatigue->Fatigue Tool*





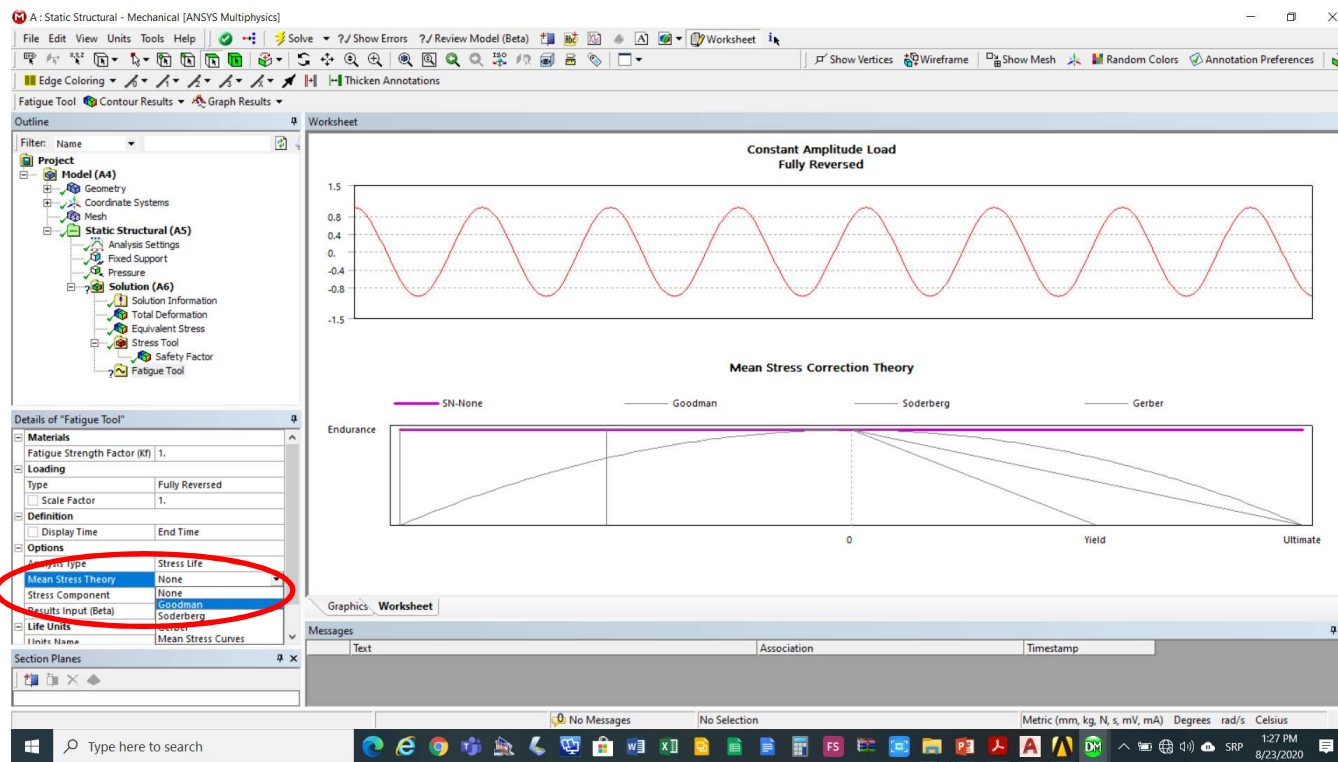
# Zamorni odkaz

Izabrati *Fully Reversed* (naizmjenično promjenljivi) tip opterećenja sa liste *Details of Fatigue Tool*->*Loading*->*Type*



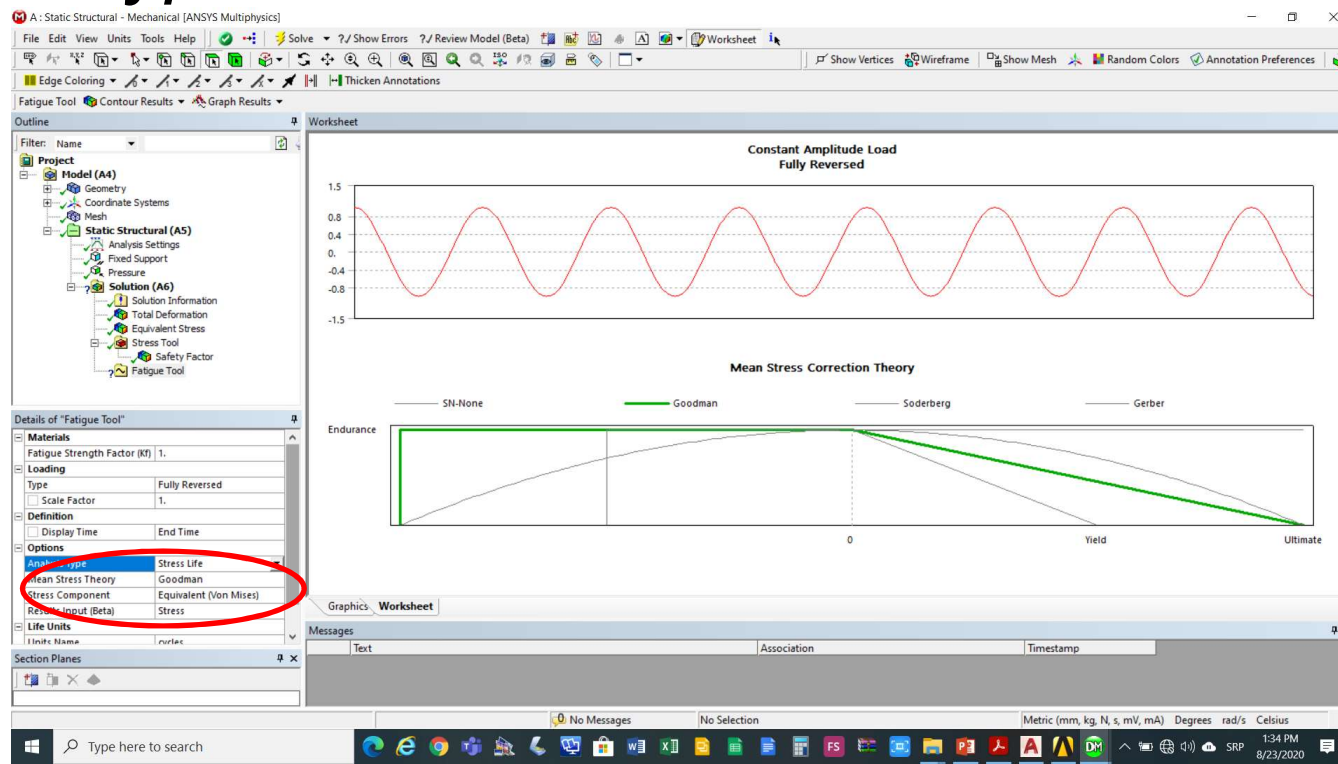
# Zamorni otkaz

Izabrati *Goodman* model za analizu zamornog otkaza sa liste *Details of Fatigue Tool*->*Options*->*Mean Stress Theory*



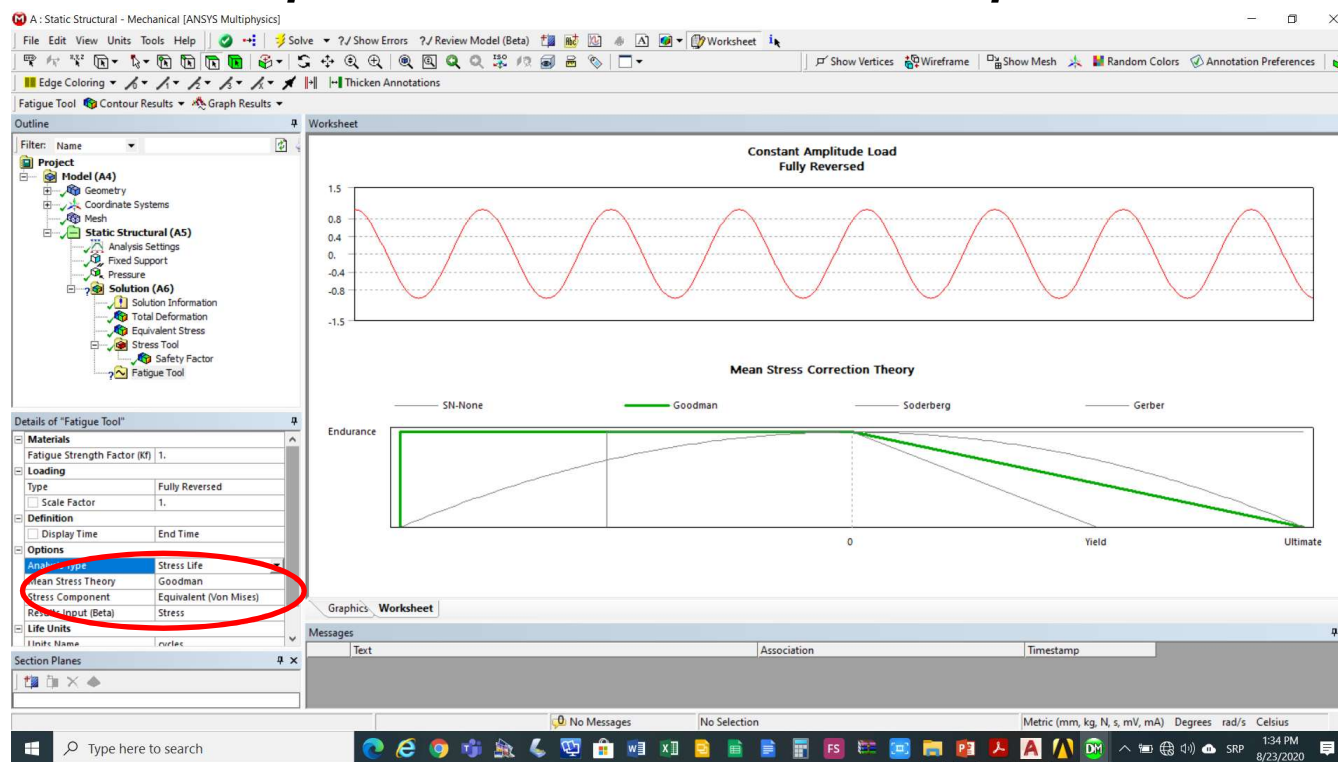
# Zamorni otkaz

Izabrati *Stress Life* proračun dužine radnog vijeka sa liste *Details of Fatigue Tool->Options->Analysis Type*



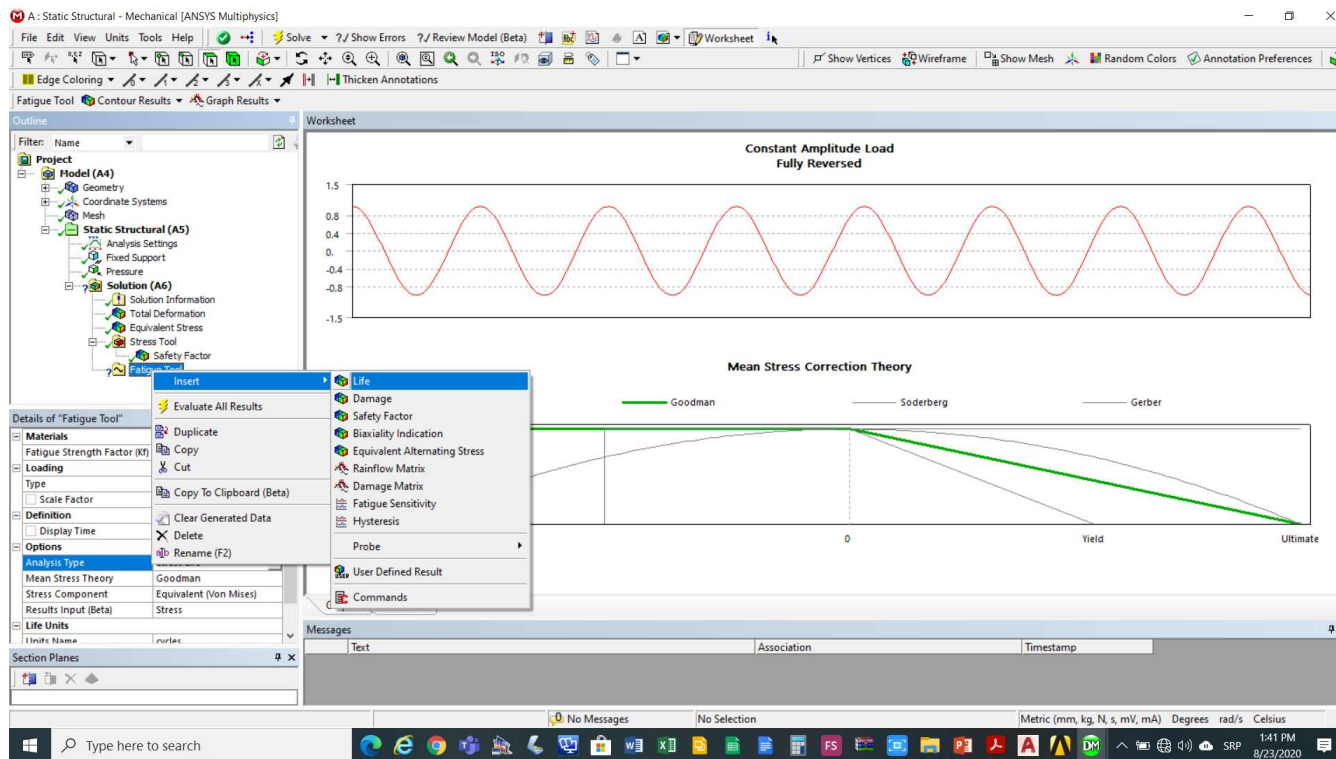
# Zamorni odkaz

Izabrati *Equivalent von Mises* napon prema kojem se vrši analiza otkaza sa liste *Details of Fatigue Tool*->*Options*->*Stress Component*



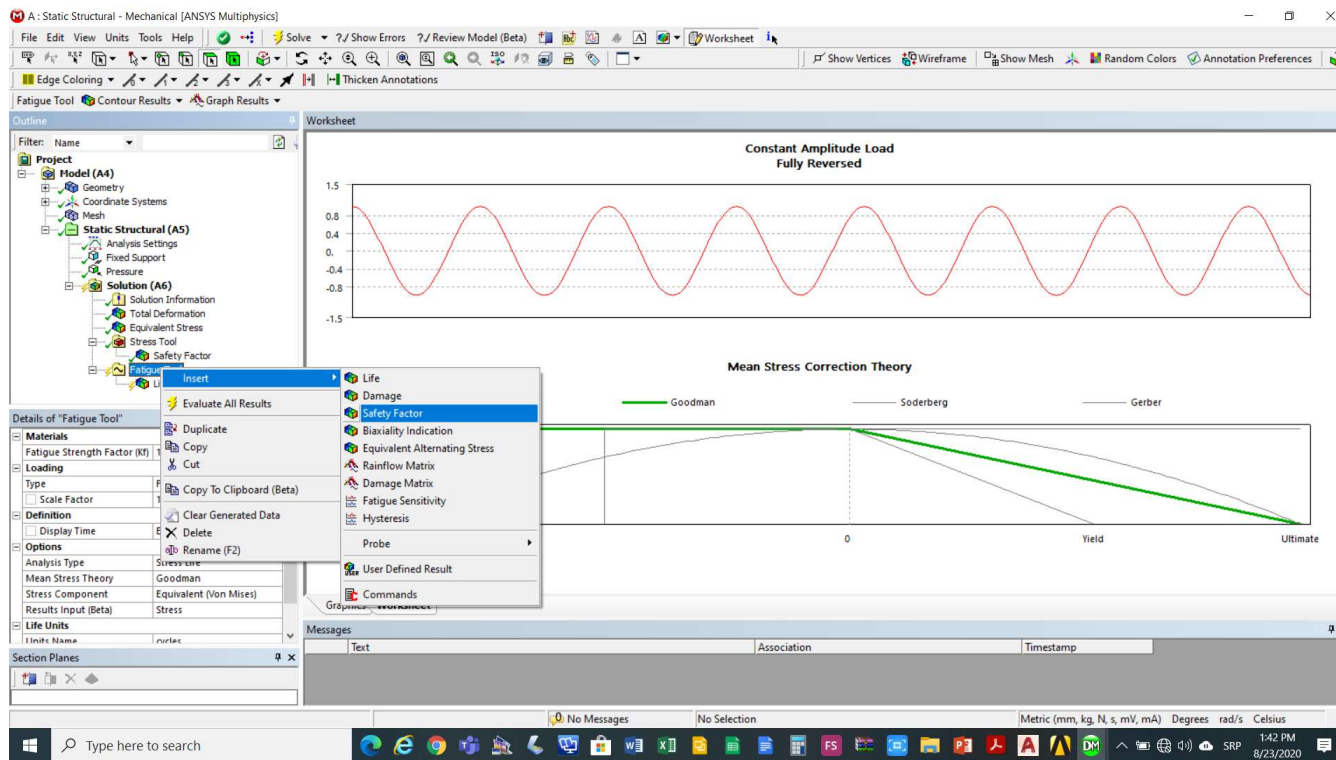
# Zamorni otkaz

Dodati određivanje dužine radnog vijeka *Fatigue Tool*->*Insert*->*Life*



# Zamorni otkaz

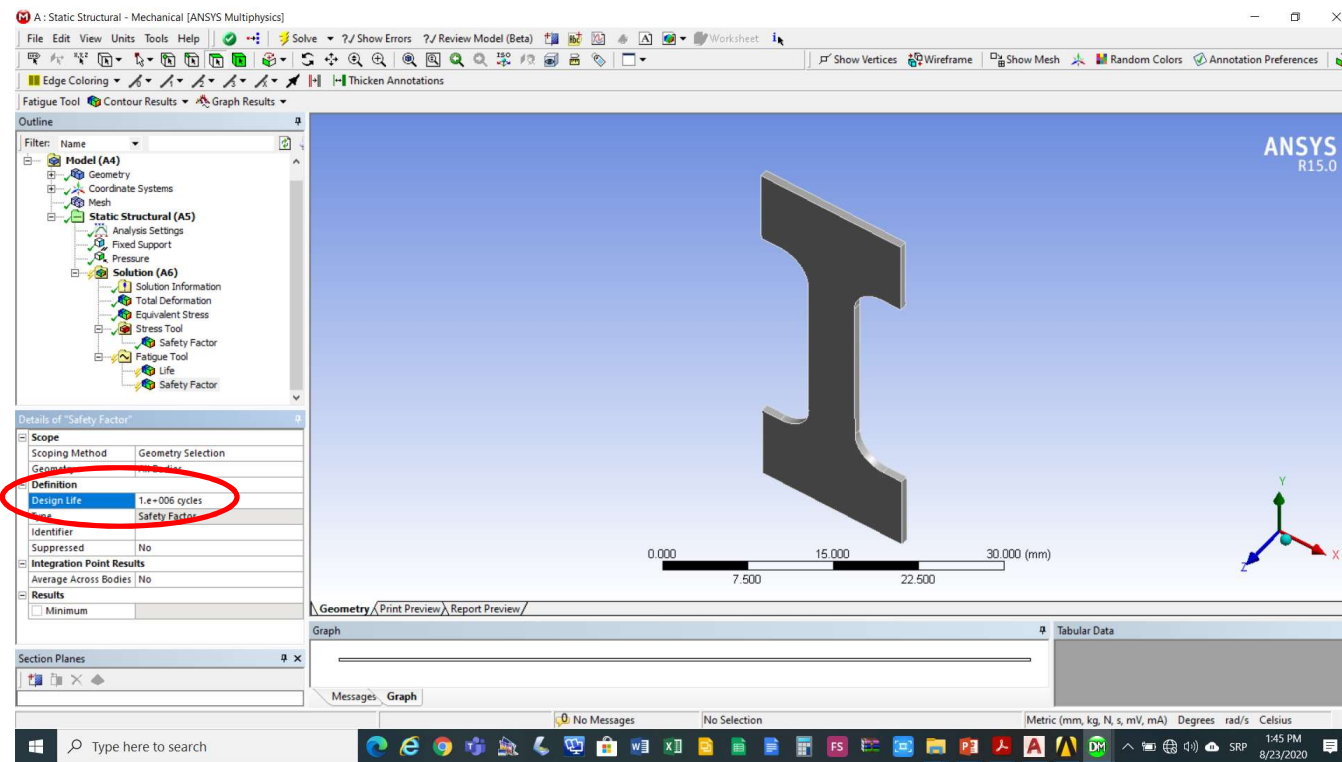
Dodati određivanje stepena sigurnosti u odnosu na zamorni otkaz *Fatigue Tool->Insert->Safety Factor*





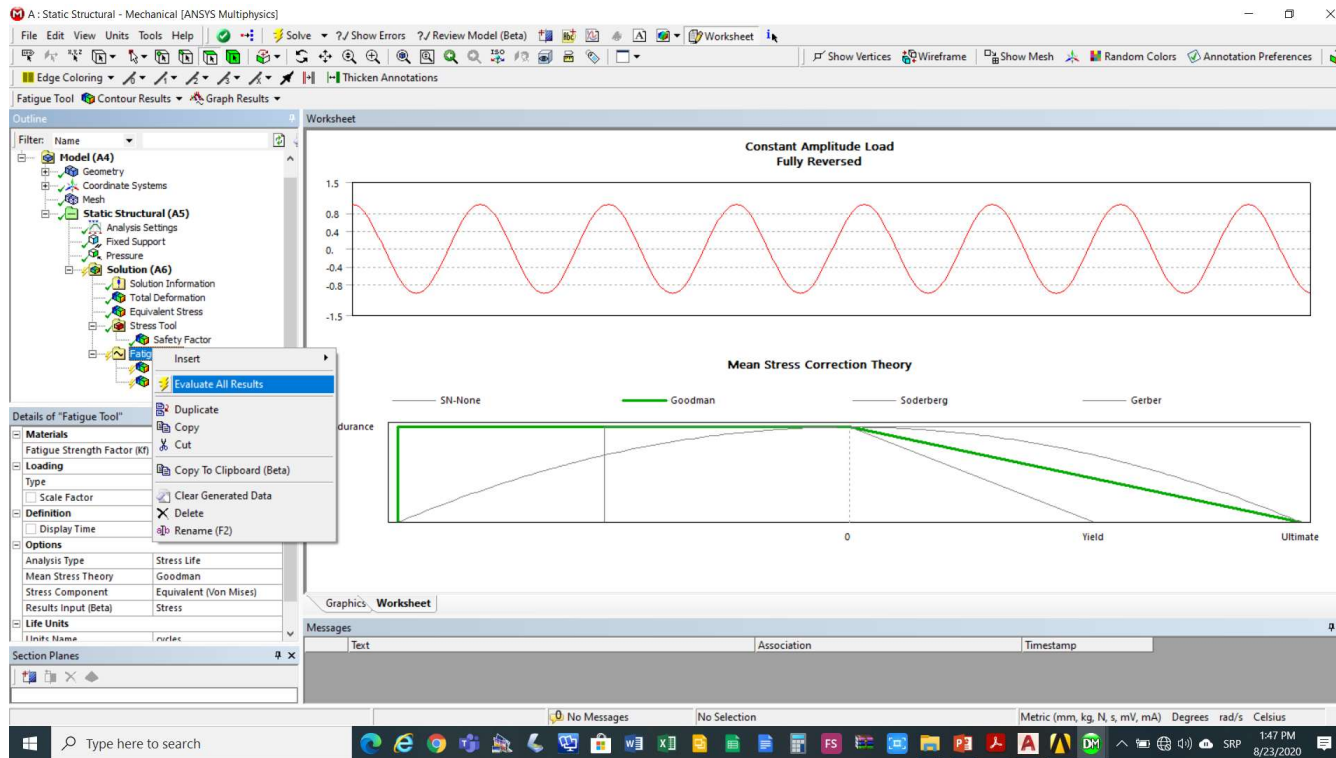
# Zamorni otkaz

Podesiti projektovanu dužinu radnog vijeka unosom  $10^6$  u polje *Deatils of Safety Factor*->*Definition*->*Design Life*



# Zamorni otkaz

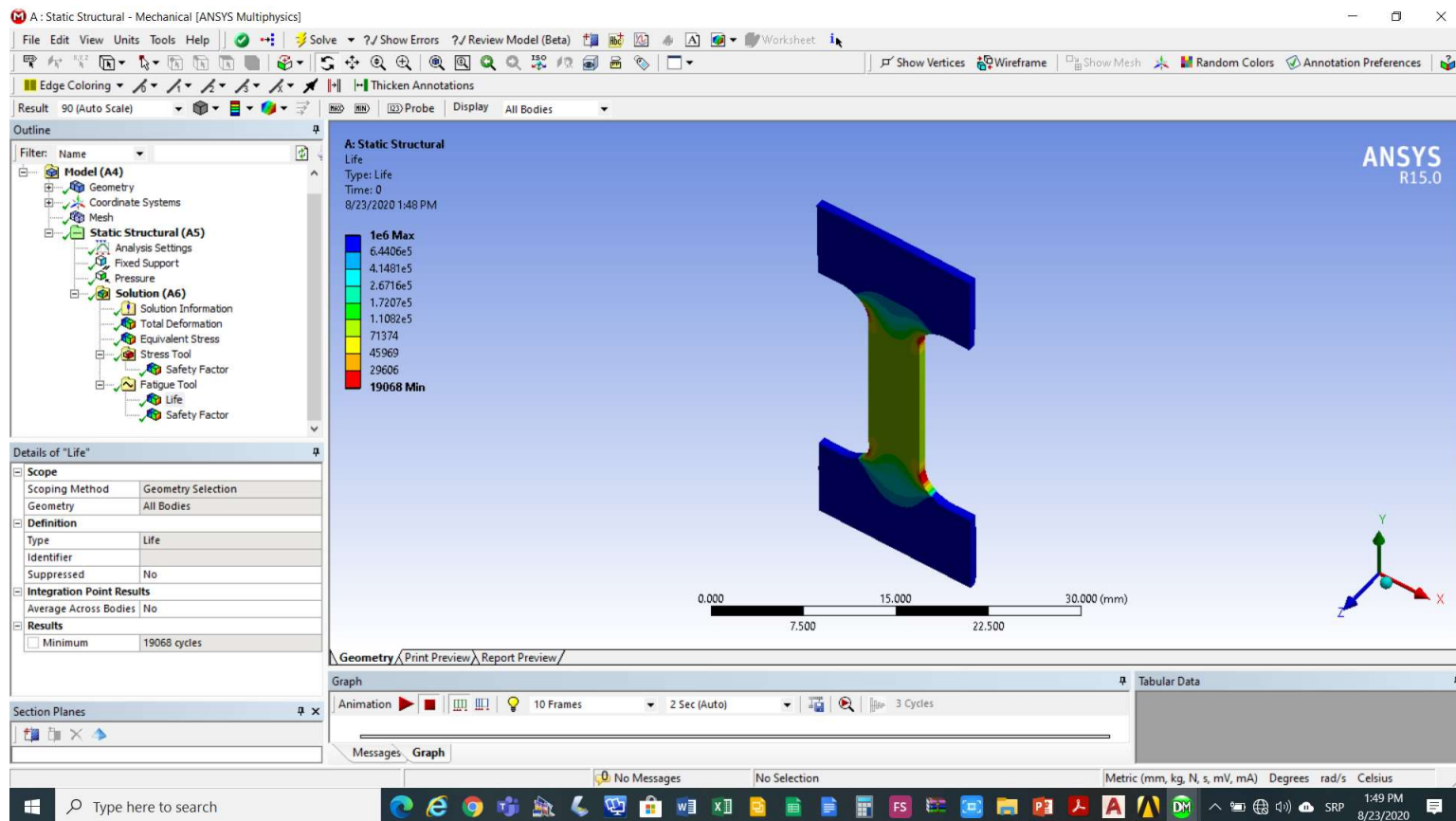
Aktivirati izvršenje analize zamora *Fatigue Tool*-  
>*Evaluate all Results*





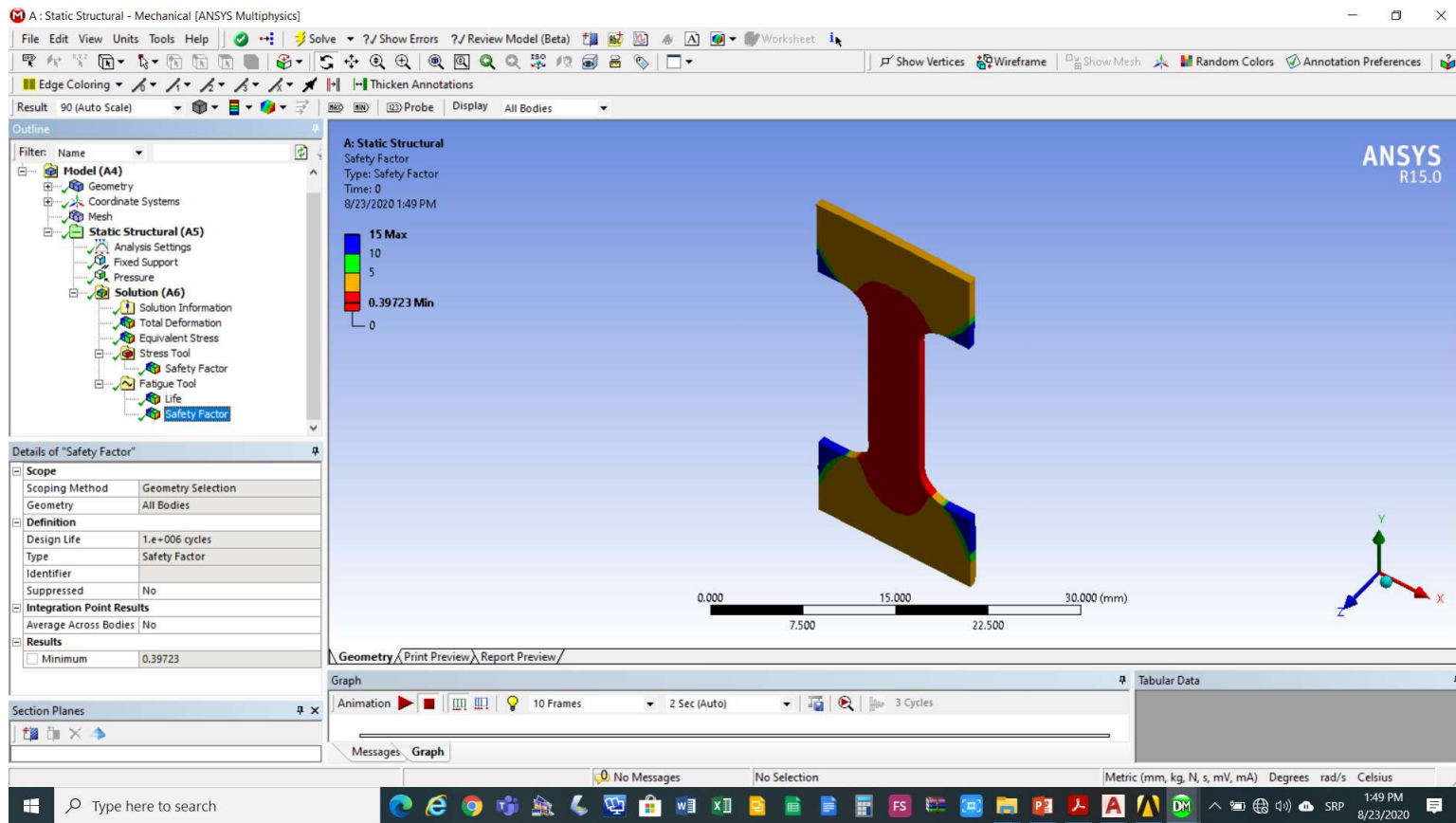
# Zamorni otkaz

## Raspodjela ciklusa do zamornog otkaza



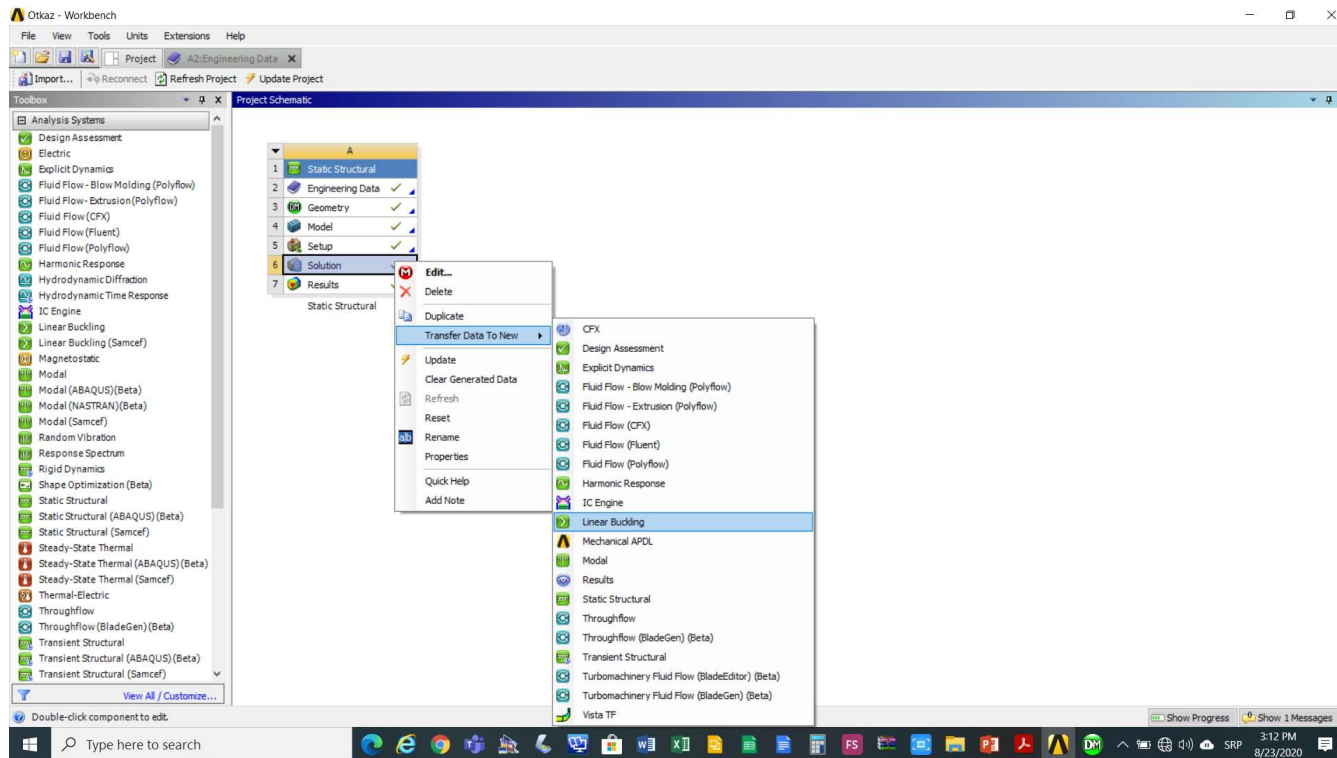
# Zamorni otkaz

## Raspodjela stepena sigurnosti protiv zamornog otkaza



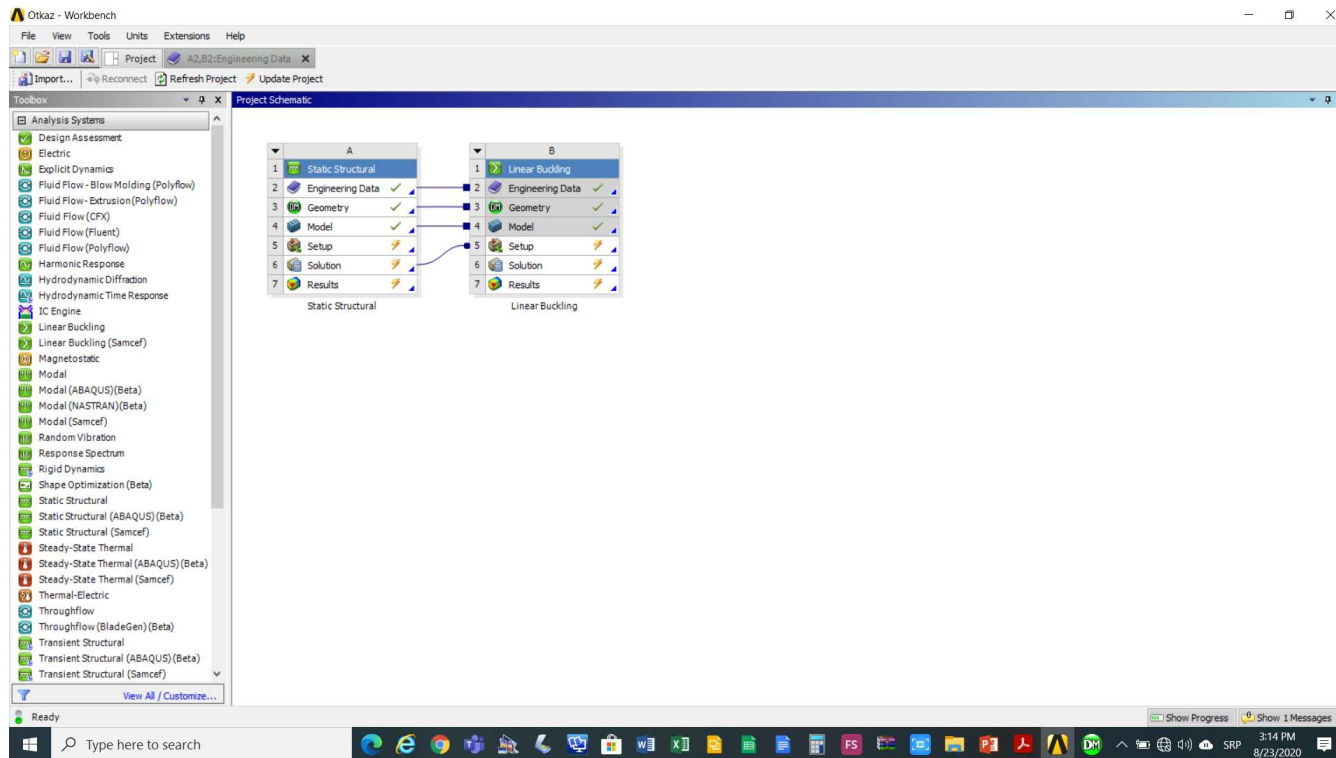
# Otkaz usled izvijanja

Kreirati linearnu analizu izvijanja *Solution-*  
> *Transfer Data to New->Linear Buckling*



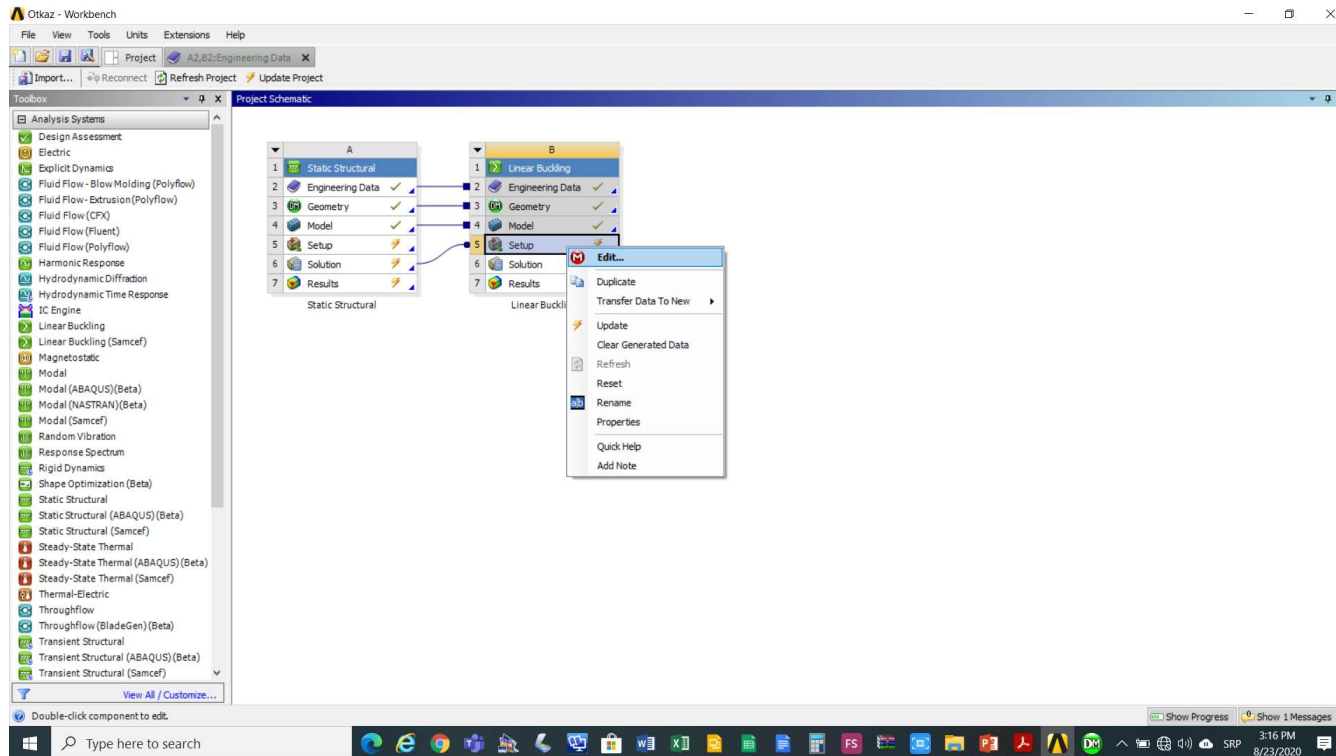
# Otkaz usled izvijanja

Kreirati linearnu analizu izvijanja *Solution-*  
> *Transfer Data to New->Linear Buckling*



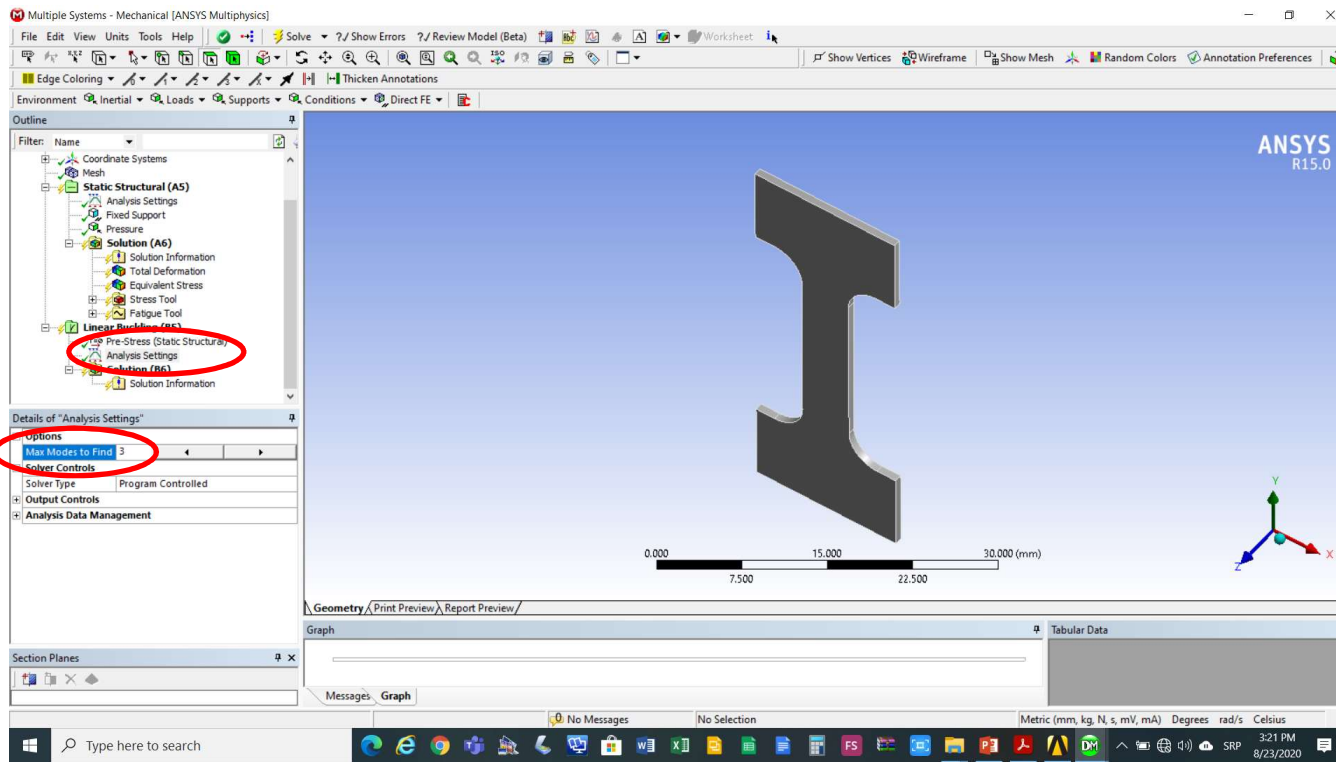
# Otkaz usled izvijanja

Aktivirati *Multiple Systems Mechanical* sa dva klika na opciju *Setup*



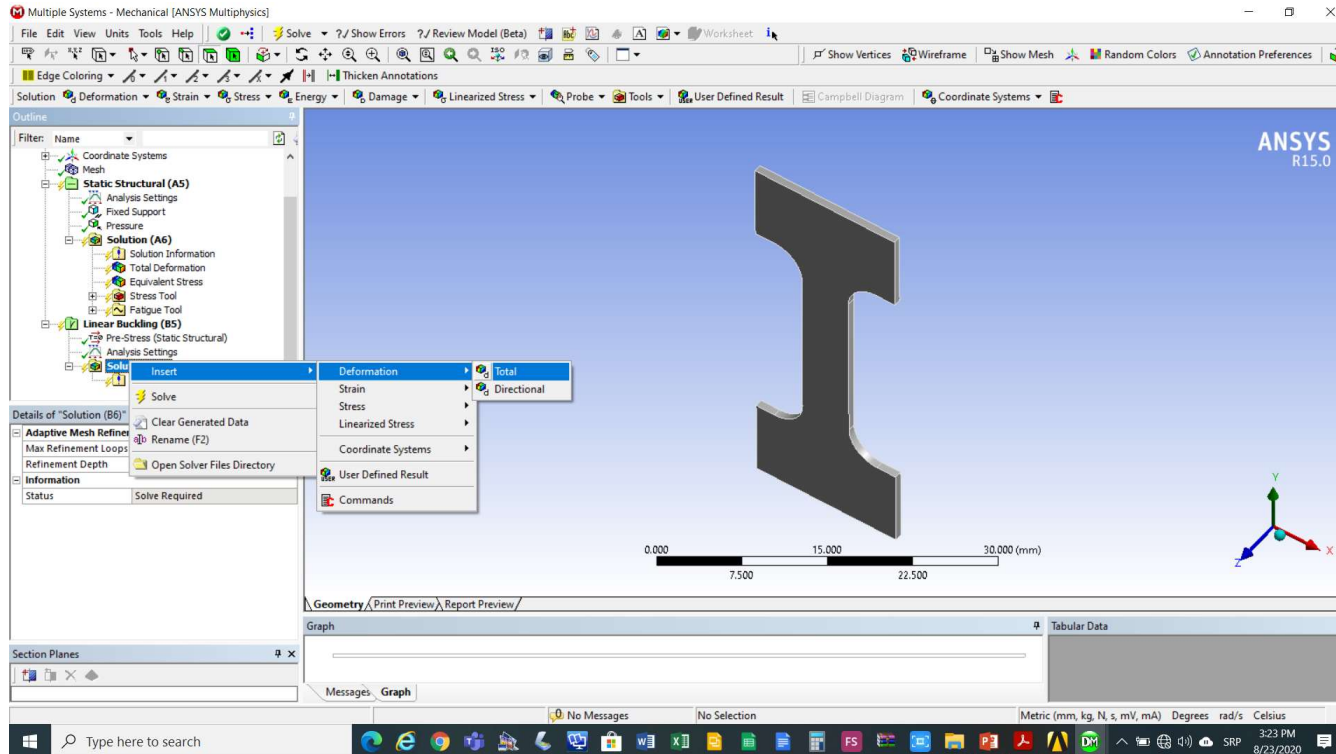
# Otkaz usled izvijanja

Podesiti broj oblika izvijanja unosom broja 3 u polje *Details of Analysis Settings->Options->Max Modes to Find*



# Otkaz usled izvijanja

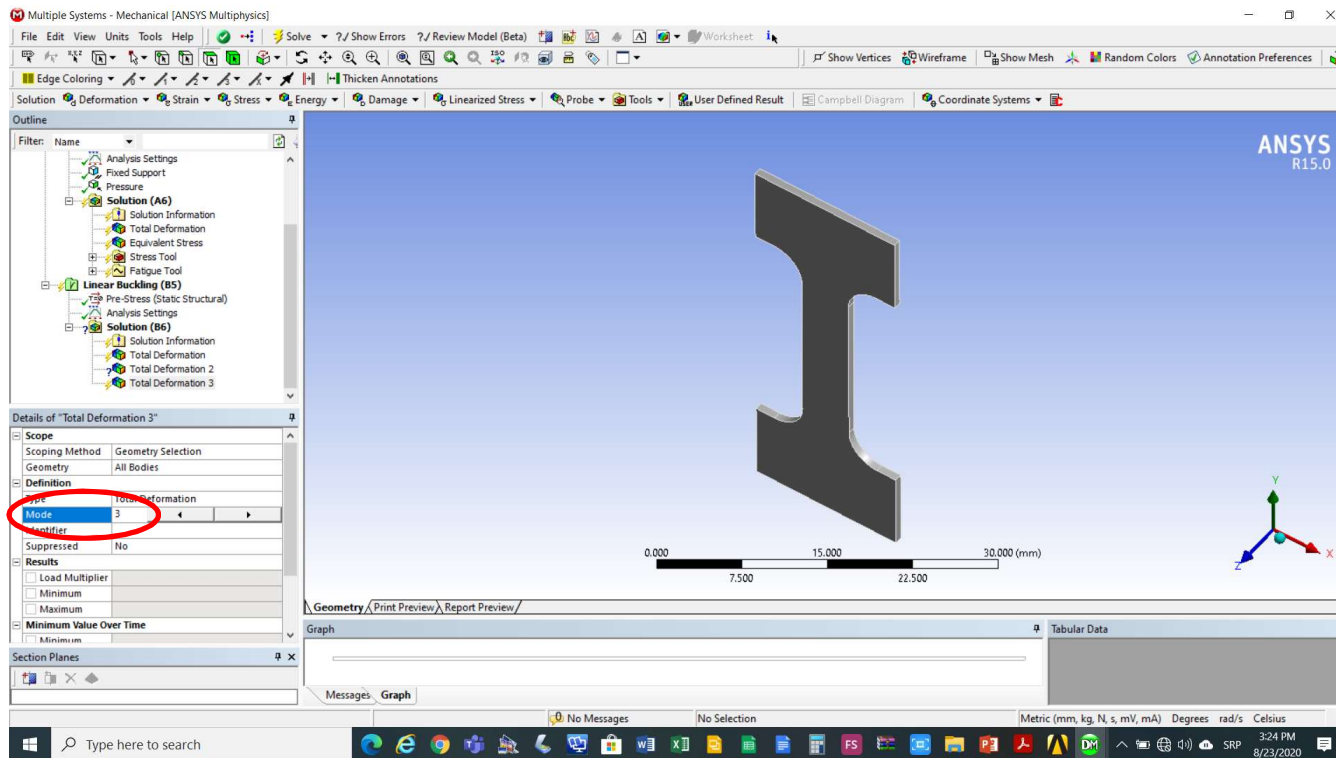
Izabrati analizu koja se želi realizovati *Solution->Insert->Total Deformation*





# Otkaz usled izvijanja

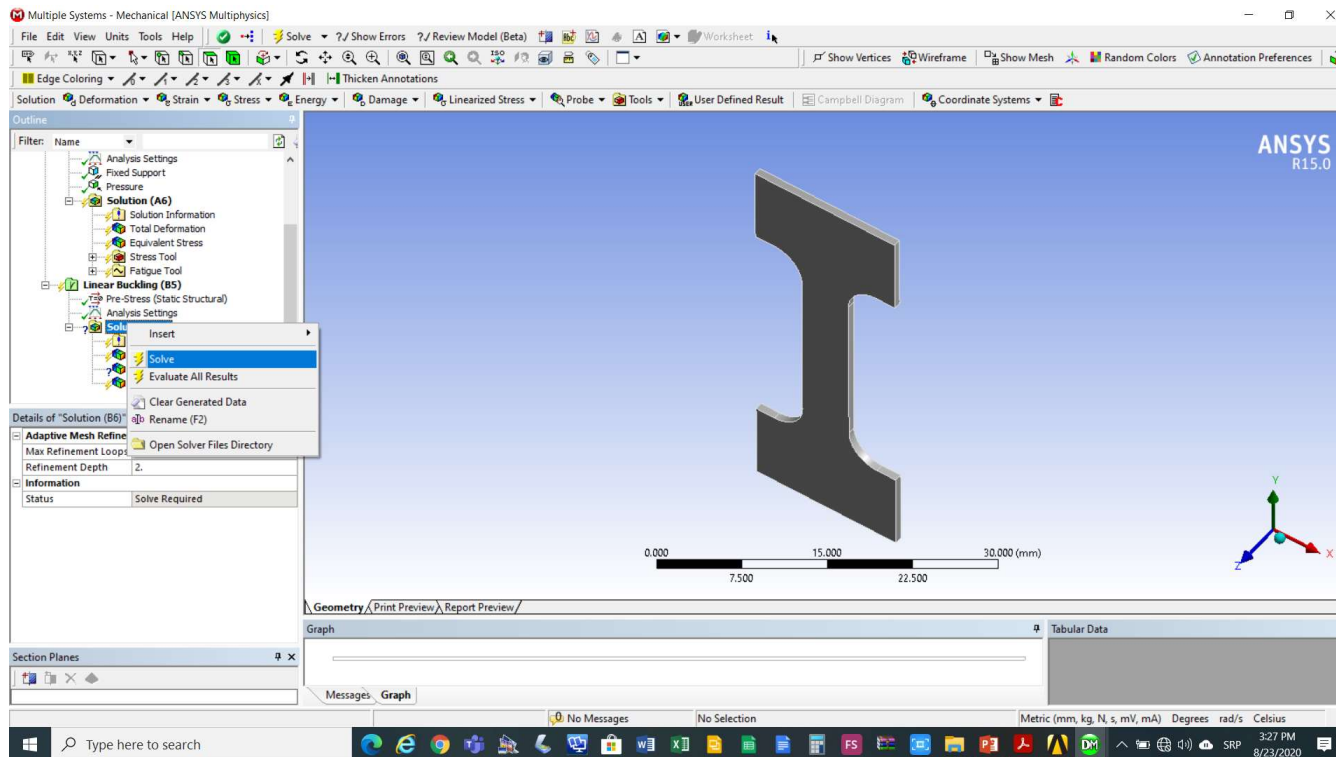
Unijeti brojeve 1, 2 i 3 u polja *Details of Total Deformation*->*Definition*->*Mode*





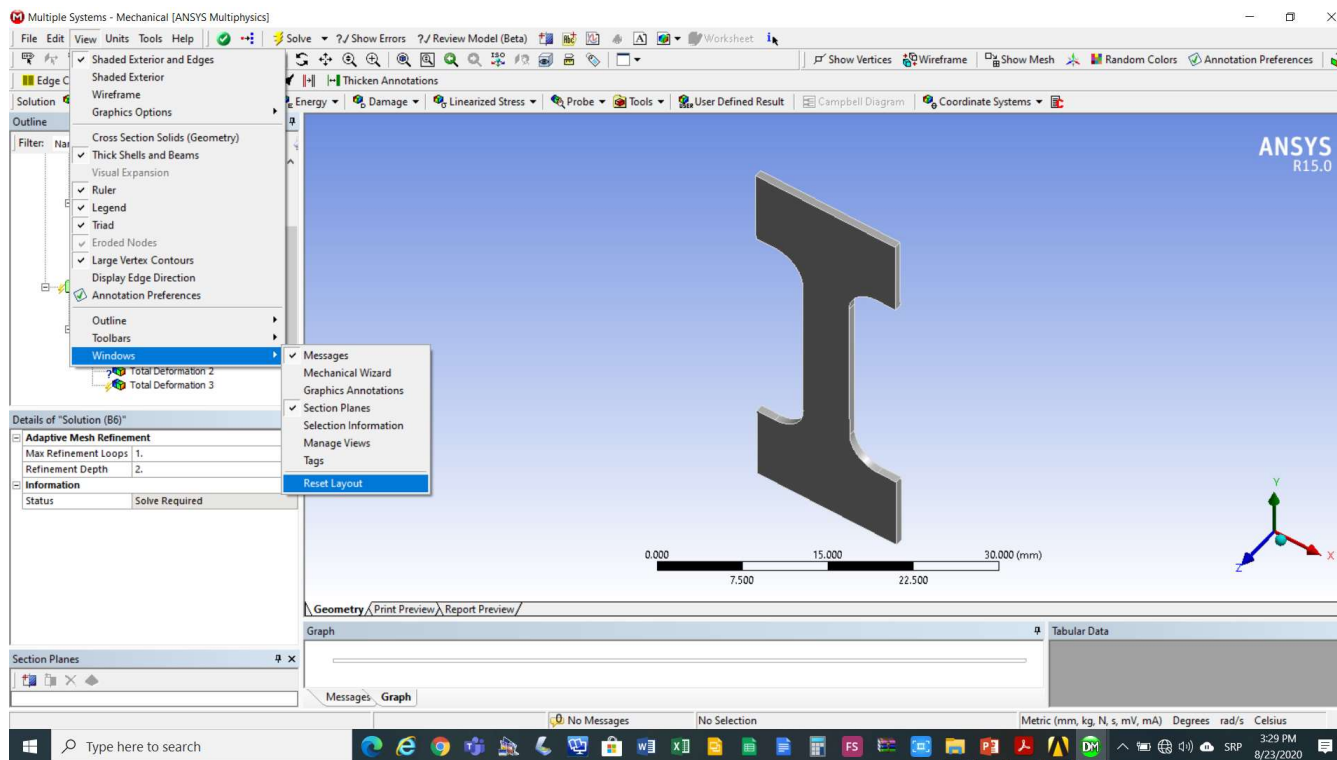
# Otkaz usled izvijanja

Aktivirati izvršenje analize *Solution*->*Solve*



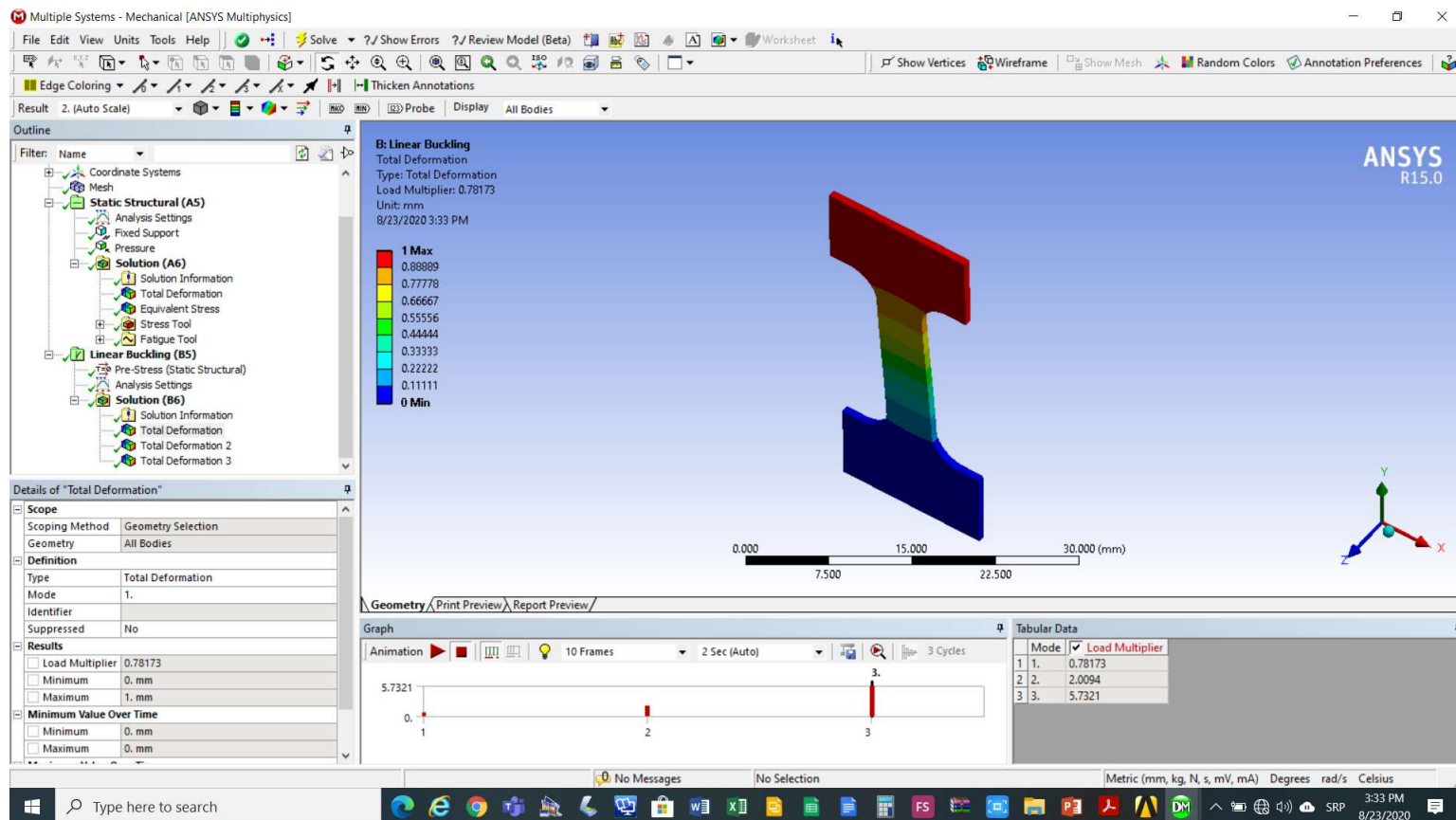
# Otkaz usled izvijanja

Aktivirati opciju sa padajućeg menija *View->Windows->Reset Layout*



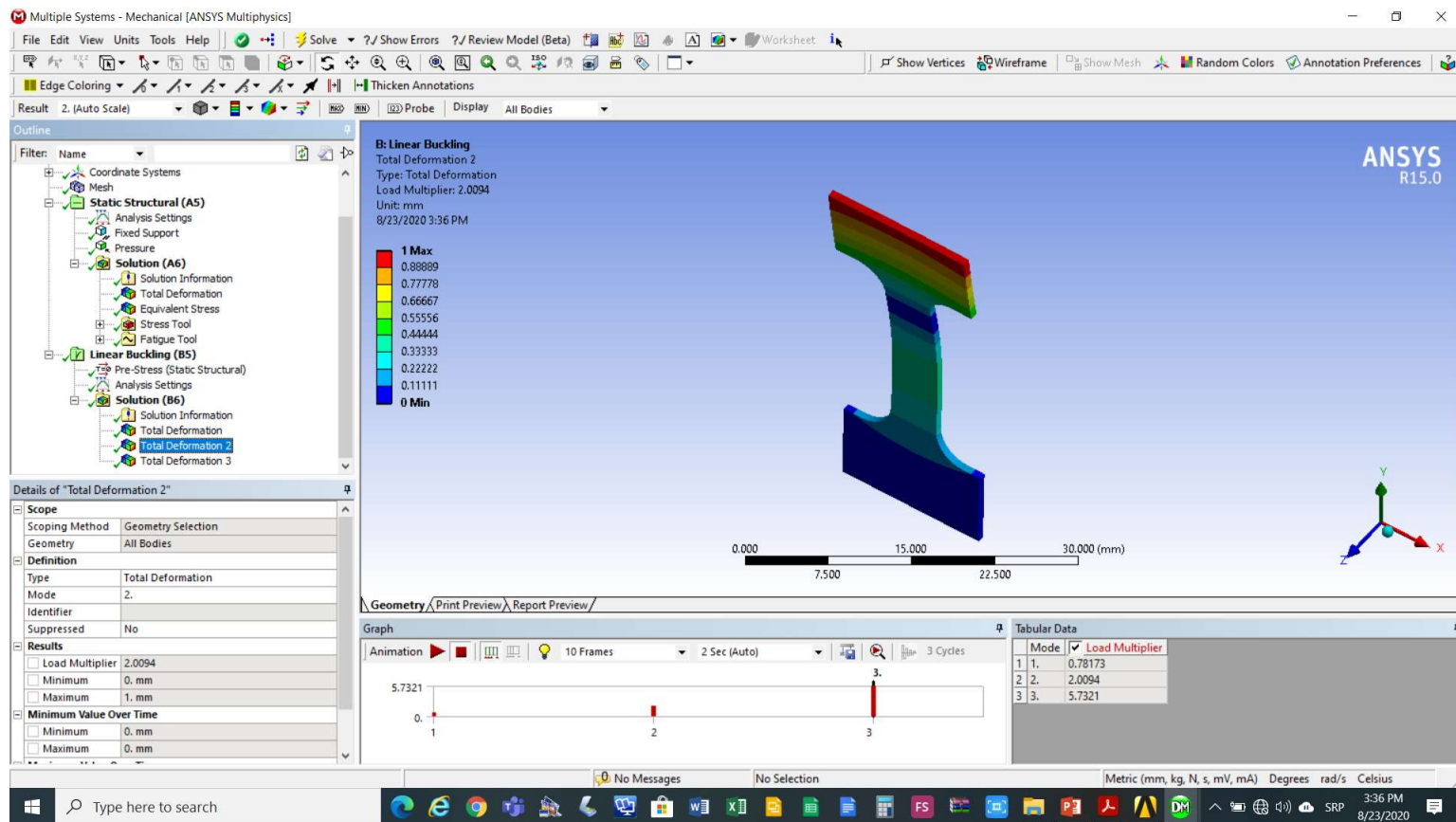
# Otkaz usled izvijanja

Prvi oblik izvijanja za pritisak  $0.78173 \cdot 50 \text{ MPa}$



# Otkaz usled izvijanja

Drugi oblik izvijanja za pritisak 2.0094·50 MPa



# Otkaz usled izvijanja

Treći oblik izvijanja za pritisak 5.7321·50 MPa

