

Nedjelja 6

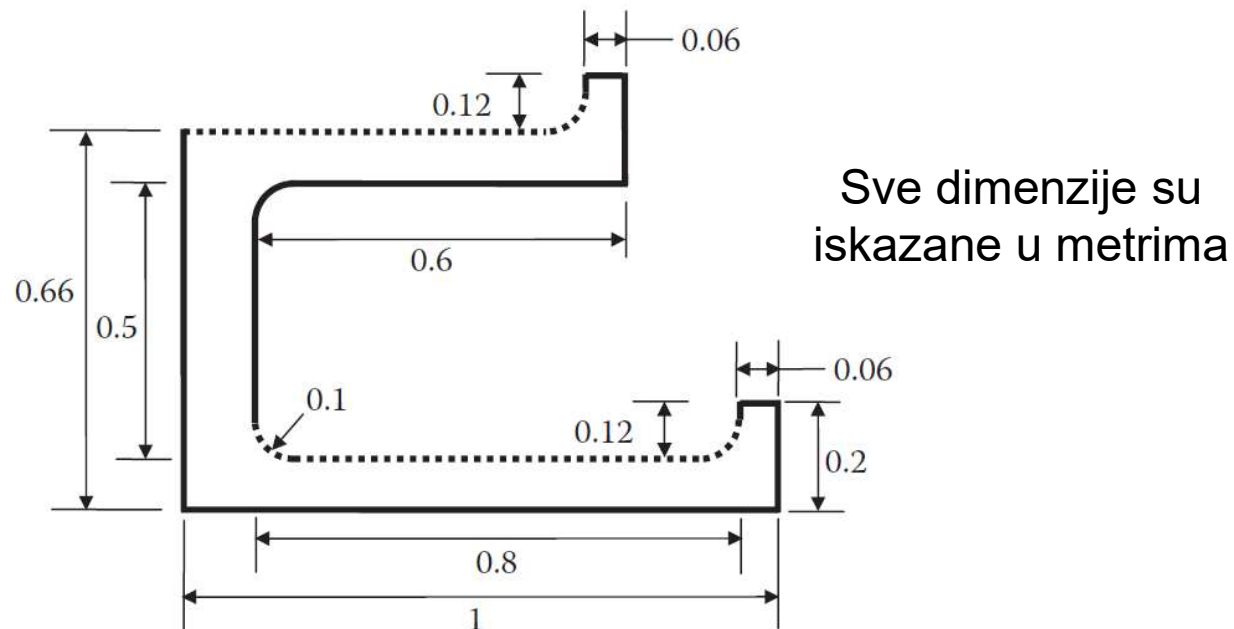
Modeliranje adaptivne mreže konačnih elmenata

Postavka zadatka

Konstrukcije baštenske fontane je izrađena od livenog željeza. Geometrija, opterećenje i oslonci su raspoređeni osno simetrično u odnosu na osu koja prolazi kroz lijevu ivicu modela prikazanog na slici. Odrediti maksimalna pomjeranja i von Mises-ove napone usled hidrostatičkog pritiska koji djeluje duž konturnih ivica prikazanih isprekidanim linijama. Oslanjanje donje ivice se modelira nepokretnim osloncima.

Postavka zadatka

2D model polovine presjeka fontane koji leži u uzdužnoj ravni koja prolazi kroz osu simetrije

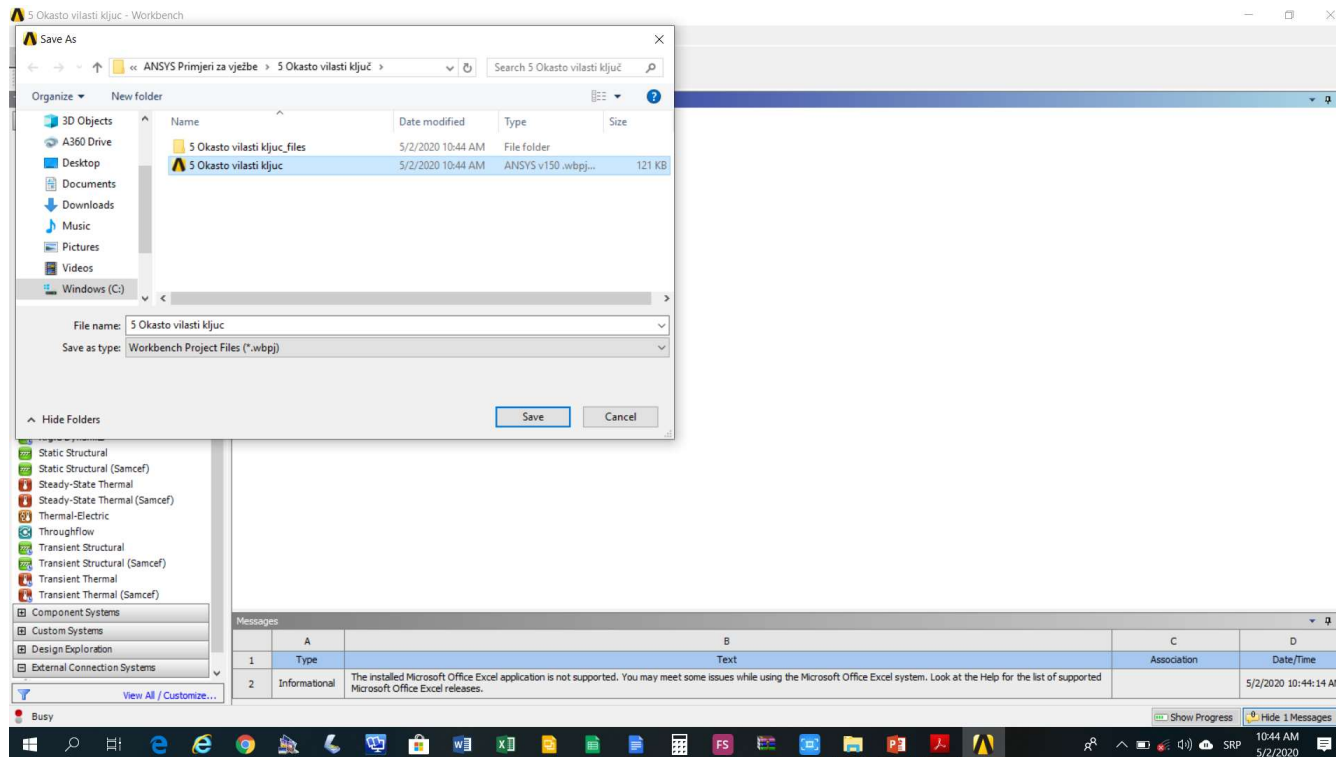


Baštenska fontana



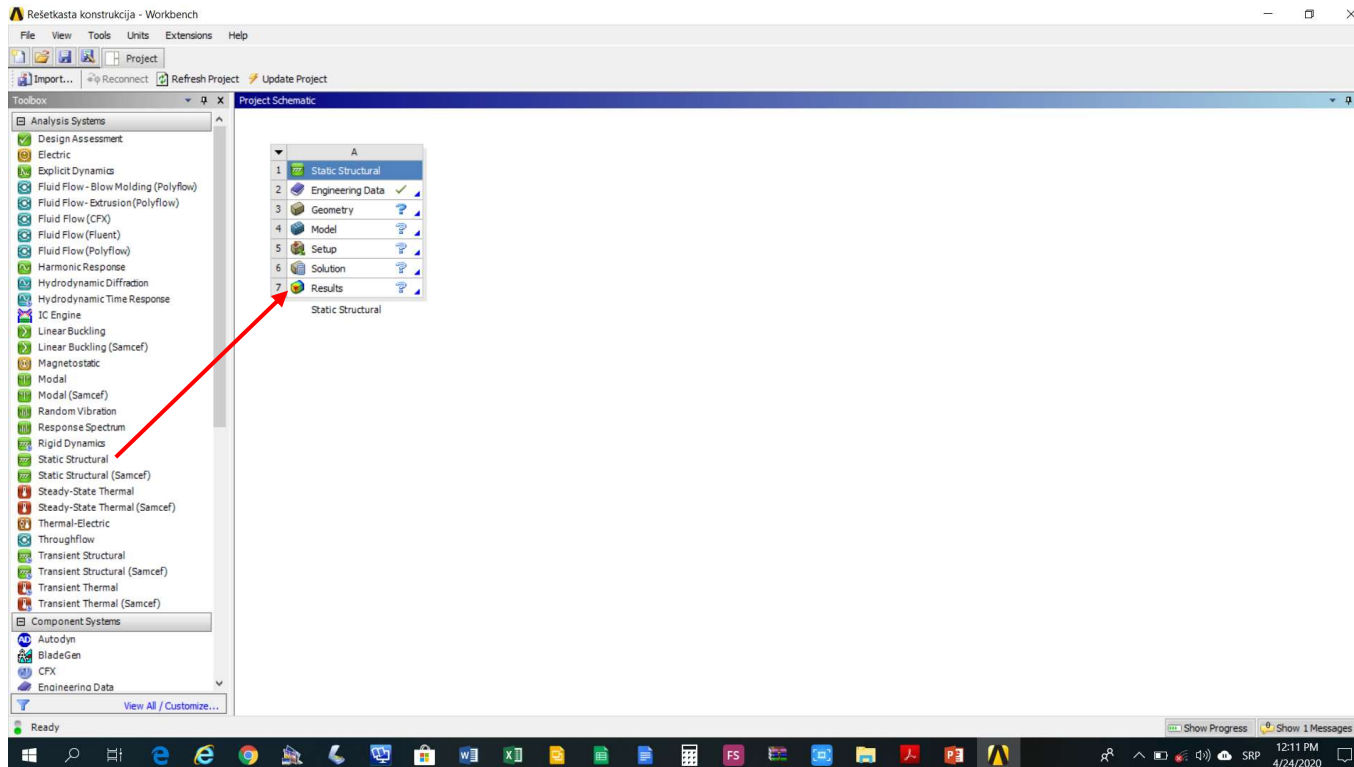
Modeliranje 2D problema

Aktivirati program ANSYS i sačuvati prazan projekat pod nazivom Fontana



Modeliranje 2D problema

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



Modeliranje 2D problema

Aktivirati modul Engineering Data (*Engineering Data->Edit*) i dodati materijale iz baze *Engineering Source Data->General Materials*

The screenshot displays the ANSYS Workbench interface. On the left, the 'Engineering Data' module is selected, and its context menu is open, showing options like 'Edit...', 'Duplicate', and 'Transfer Data From New'. The main window shows the 'Engineering Data Sources' table, where 'General Materials' is highlighted. Below this, the 'Outline of General Materials' table lists various materials such as Air, Aluminum Alloy, Concrete, Copper Alloy, Gray Cast Iron, Magnesium Alloy, Polyethylene, Silicon Anisotropic, Stainless Steel, and Structural Steel. On the right, a 'Table of Properties Row 2: Density' is shown, with a value of 7850 for Density (kg m⁻³) at 0°C. Below the table is a 'Chart of Properties Row 2: Density' showing a constant density of 7850 kg m⁻³ across a temperature range from -1 to 1°C.

1	Data Source	Location	Description
1	General		Quick access list and default items
2	General Materials		General use material samples for use in various analyses.
4	General Non-linear Materials		General use material samples for use in non-linear analyses.
5	Explicit Materials		Material samples for use in an explicit analysis.
6	Hyperelastic Materials		Material stress-strain data samples for curve fitting.
7	Magnetic B-H Curves		B-H Curve samples specific for use in a magnetic analysis.
8	Thermal Materials		Material samples specific for use in a thermal analysis.
9	Fluid Materials		Material samples specific for use in a fluid analysis.
*	Click here to add a new library		

1	Contents of General Materials	Add	Source	Description
3	Air			General properties for air.
4	Aluminum Alloy			General aluminum alloy. Fatigue properties come from MIL-HDBK-5H, page 3-277.
5	Concrete			
6	Copper Alloy			
7	Gray Cast Iron			
8	Magnesium Alloy			
9	Polyethylene			
10	Silicon Anisotropic			
11	Stainless Steel			
12	Structural Steel			Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1

1	Temperature (C)	Density (kg m ⁻³)
1	Temperature (C)	Density (kg m ⁻³)
2		7850

Modeliranje 2D problema

Učitati karakteristike livenog željeza (*Gray Cast Iron*) potom aktivirati opciju *Refresh*

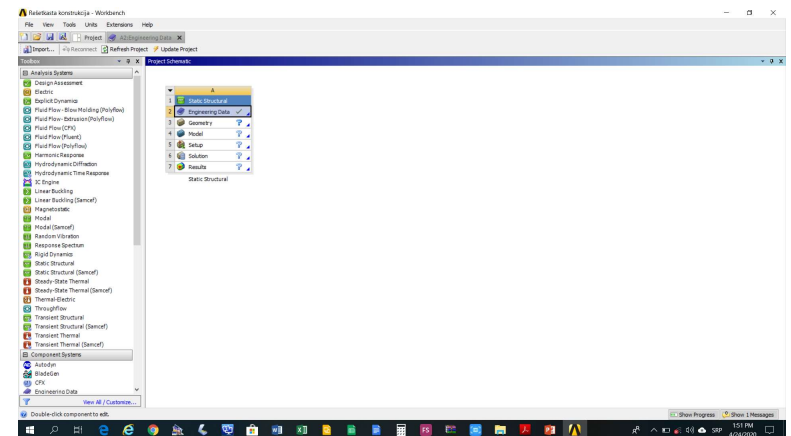
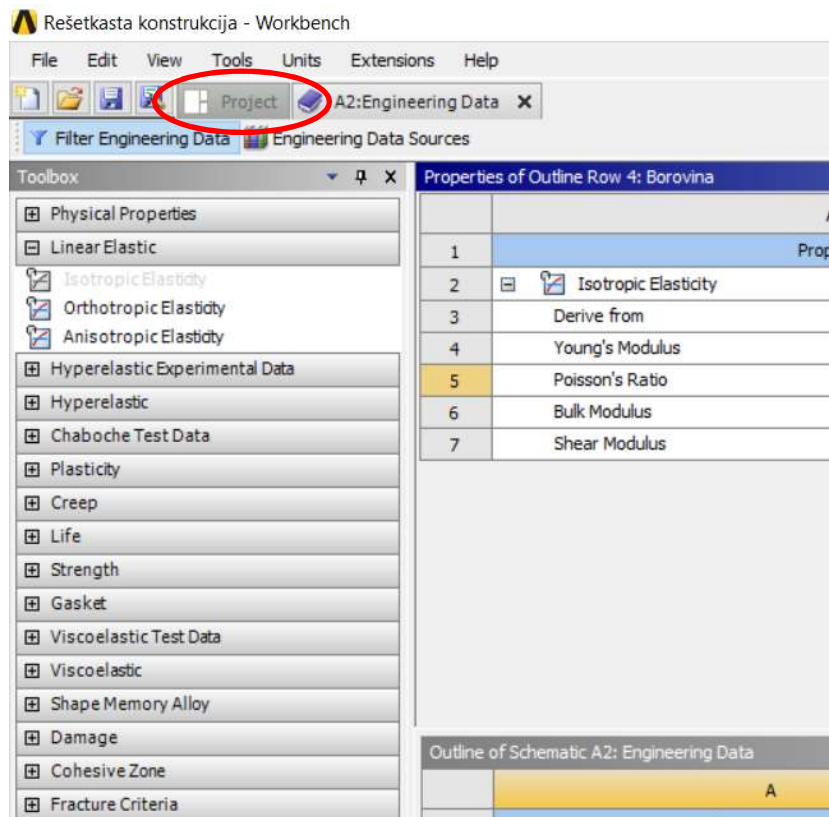
The screenshot displays the ANSYS Workbench interface. The 'Engineering Data Sources' panel is open, showing a list of material sources. 'Gray Cast Iron' is highlighted with a red circle. The 'Properties of Outline Row 4: Structural Steel' table is visible, showing the density of Gray Cast Iron as 7850 kg m⁻³. The 'Table of Properties Row 2: Density' table shows the density value of 7850. The 'Chart of Properties Row 2: Density' shows a plot of Density (kg m⁻³) versus Temperature (C). The 'View' menu is open, and the 'Refresh' option (F5) is highlighted with a red circle.

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

Properties of Outline Row 4: Structural Steel	A	B	C
1	Property	Value	Unit
2	Density	7850	kg m ⁻³
3	Isotropic Secant Coefficient of Thermal Expansion		
6	Isotropic Elasticity		
7	Derive from	Young's Modulus and Po...	
8	Young's Modulus	2E+11	Pa
9	Poisson's Ratio	0.3	
10	Bulk Modulus	1.6667E+11	Pa
11	Shear Modulus	7.6923E+10	Pa
12	Messages		

Modeliranje 2D problema

Izabrati opciju *Project* za povratak na shemu projekta



Modeliranje 2D problema

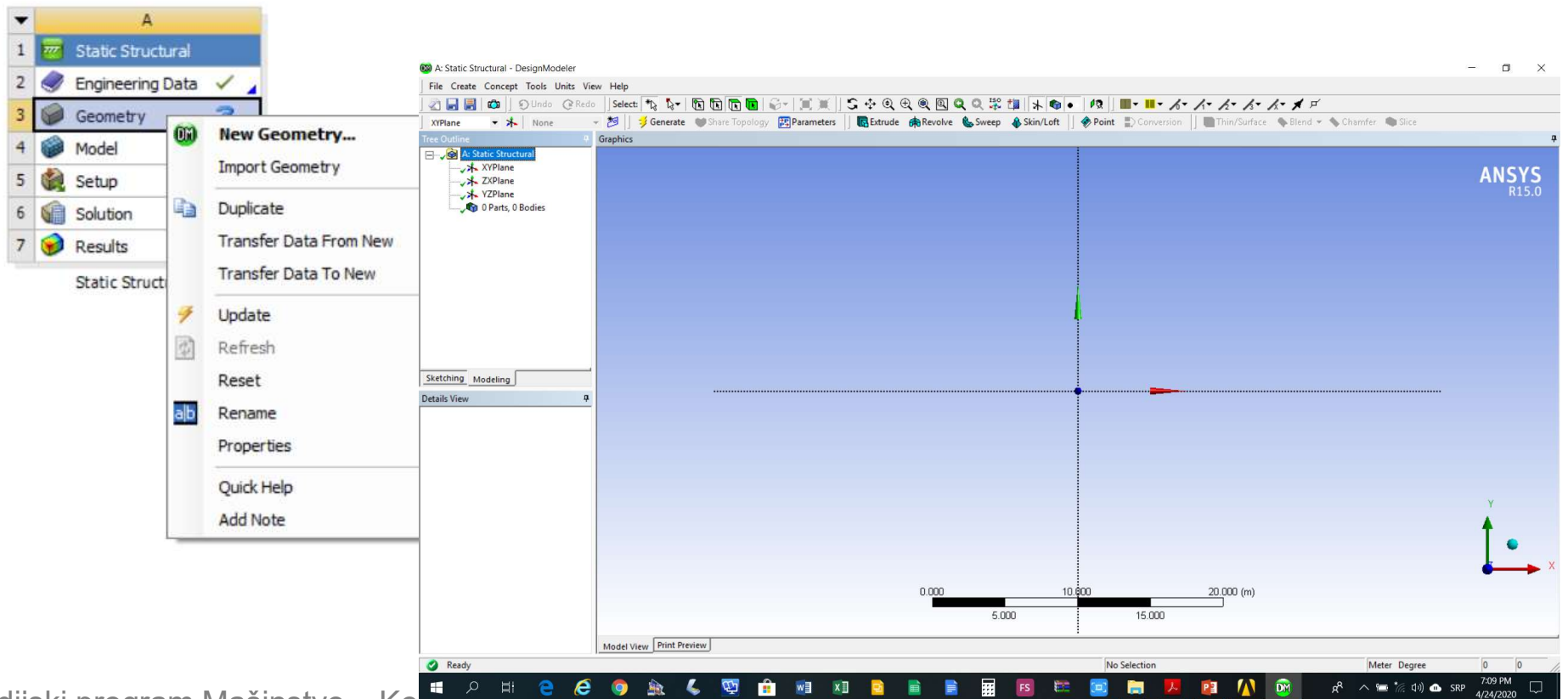
Izvršiti podešavanja modula Design Modeler
(*Geometry*->*Properties*->*Analysis Type* = 2D)

The screenshot displays the ANSYS Workbench interface. On the left, the 'Properties' dialog for the 'Geometry' component is open, showing various options. The 'Analysis Type' is set to '2D', and 'Surface Bodies' is checked. The 'Properties' dialog is divided into several sections: General, Notes, Used Licenses, Geometry Source, Basic Geometry Options, and Advanced Geometry Options. The 'Advanced Geometry Options' section is highlighted with a red circle, and the 'Analysis Type' is set to '2D'. The 'Basic Geometry Options' section is also highlighted with a red circle, and 'Surface Bodies' is checked. The 'Properties' dialog is also divided into two columns, A and B, with 'Property' and 'Value' headers. The 'Value' column contains the following data:

Property	Value
Component ID	Geometry
Directory Name	SYS
Geometry File Name	
Surface Bodies	<input checked="" type="checkbox"/>
Line Bodies	<input type="checkbox"/>
Parameters	<input checked="" type="checkbox"/>
Parameter Key	DS
Attributes	<input type="checkbox"/>
Named Selections	<input type="checkbox"/>
Material Properties	<input type="checkbox"/>
Analysis Type	2D
Use Associativity	<input type="checkbox"/>
Use Coordinate Systems	<input type="checkbox"/>
Import Work Points	<input type="checkbox"/>
Reader Mode Saves Updated File	<input type="checkbox"/>
Import Using Instances	<input checked="" type="checkbox"/>
Smart CAD Update	<input type="checkbox"/>
Compare Parts On Update	No
Enclosure and Symmetry Processing	<input checked="" type="checkbox"/>
Decompose Disjoint Geometry	<input checked="" type="checkbox"/>
Mixed Import Resolution	None

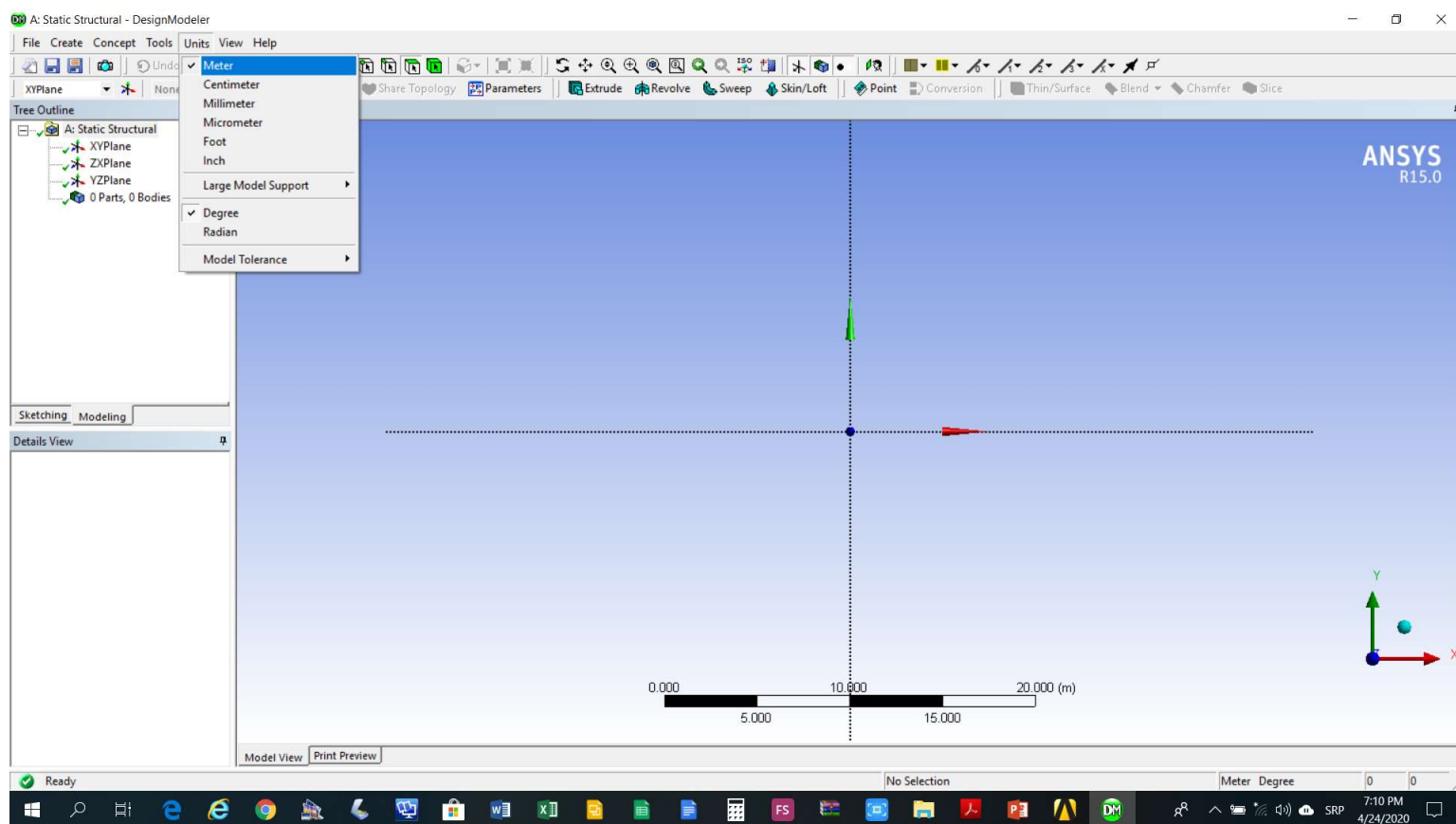
Modeliranje 2D problema

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



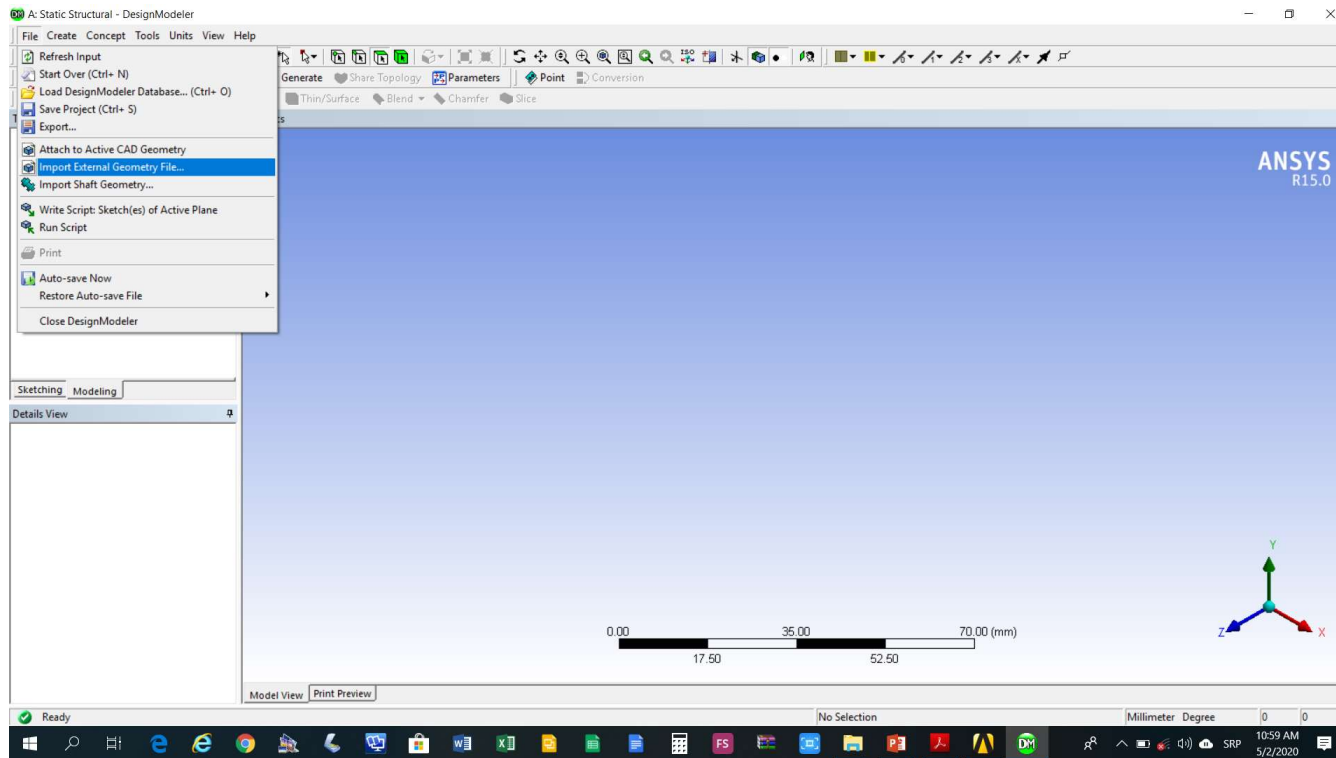
Modeliranje 2D problema

Podesiti dužinske jedinice (Units->Meter)



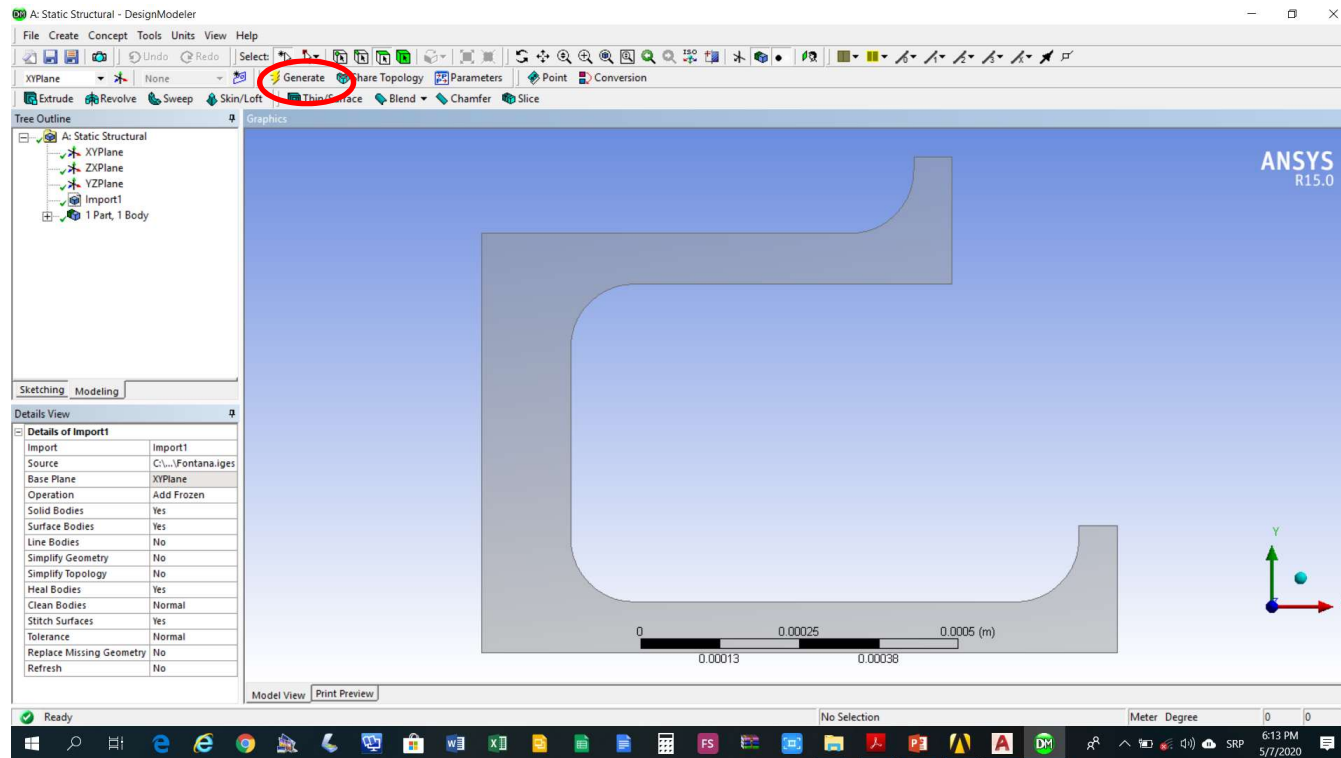
Modeliranje 2D problema

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



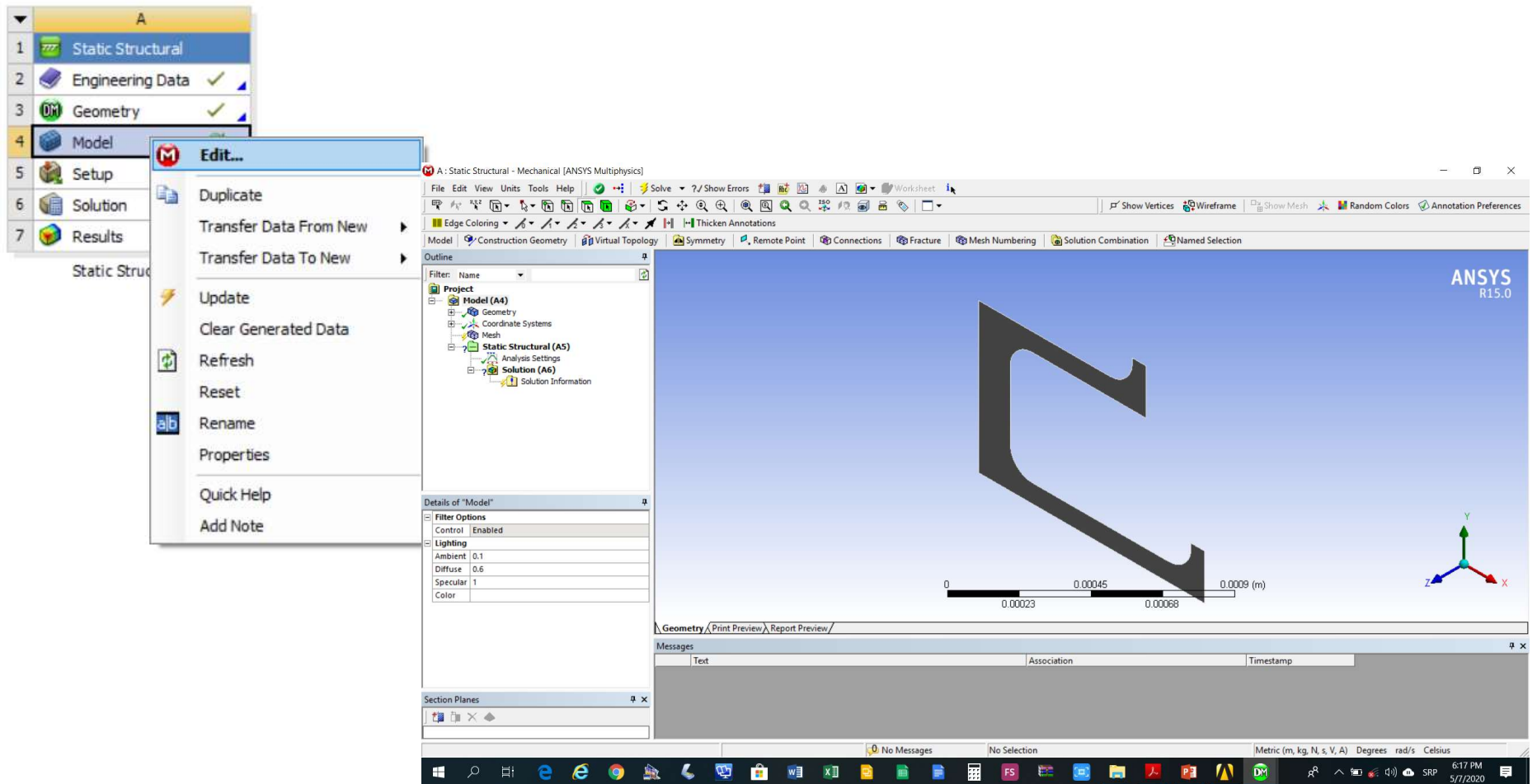
Modeliranje 2D problema

Učitavanje eksterno generisane geometrije
okončati komandom *Generate*



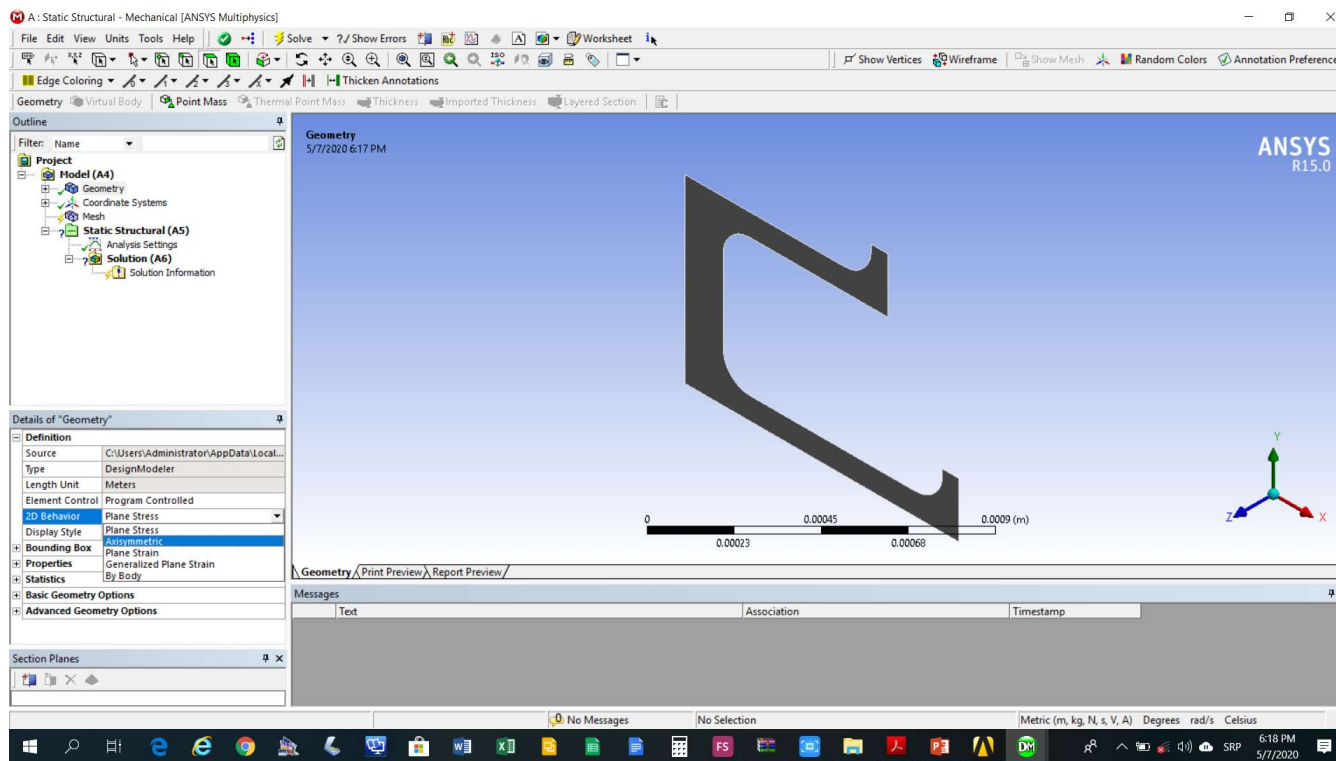
Modeliranje 2D problema

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



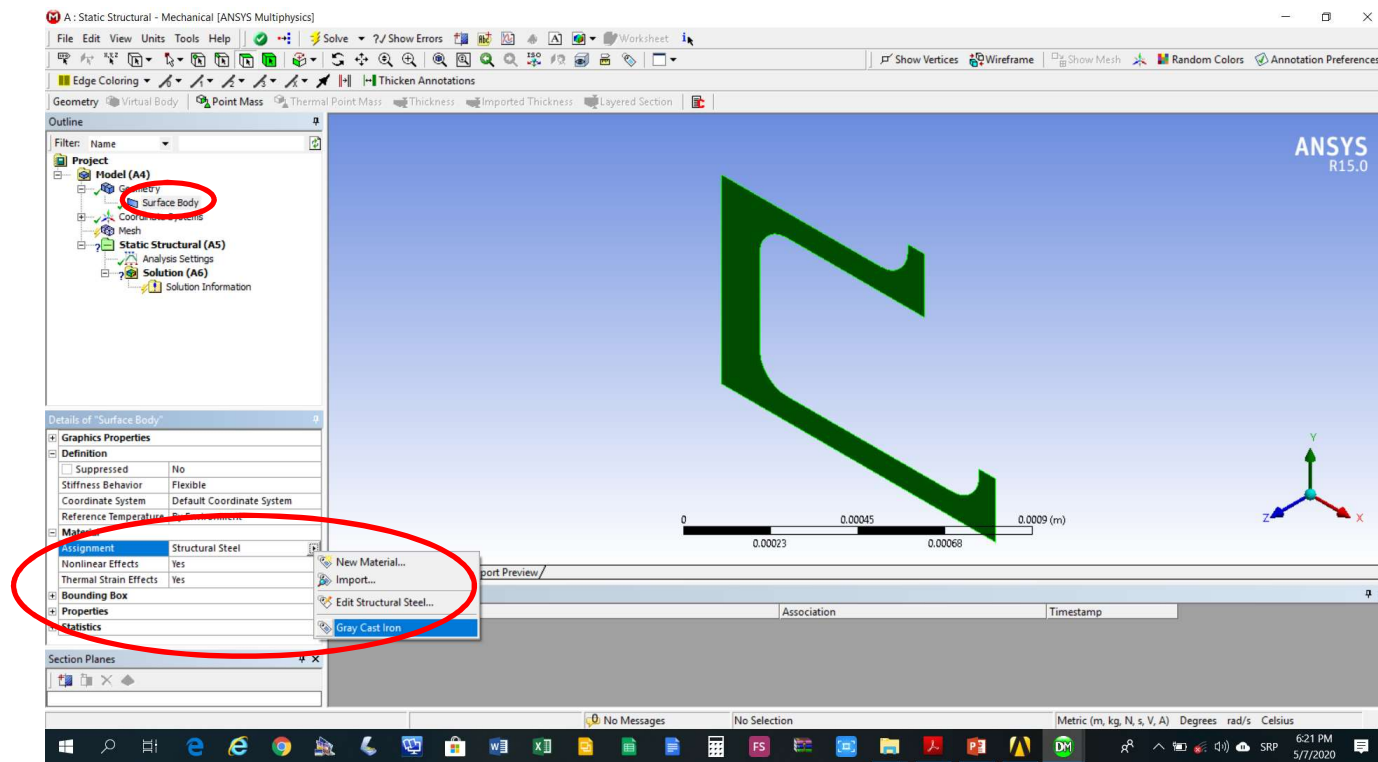
Modeliranje 2D problema

Izabrati osnosimetričnu analizu ravanskog stanja napona sa liste *Details of Geometry->2D Behaviour* izabrati *Axisymmetric*



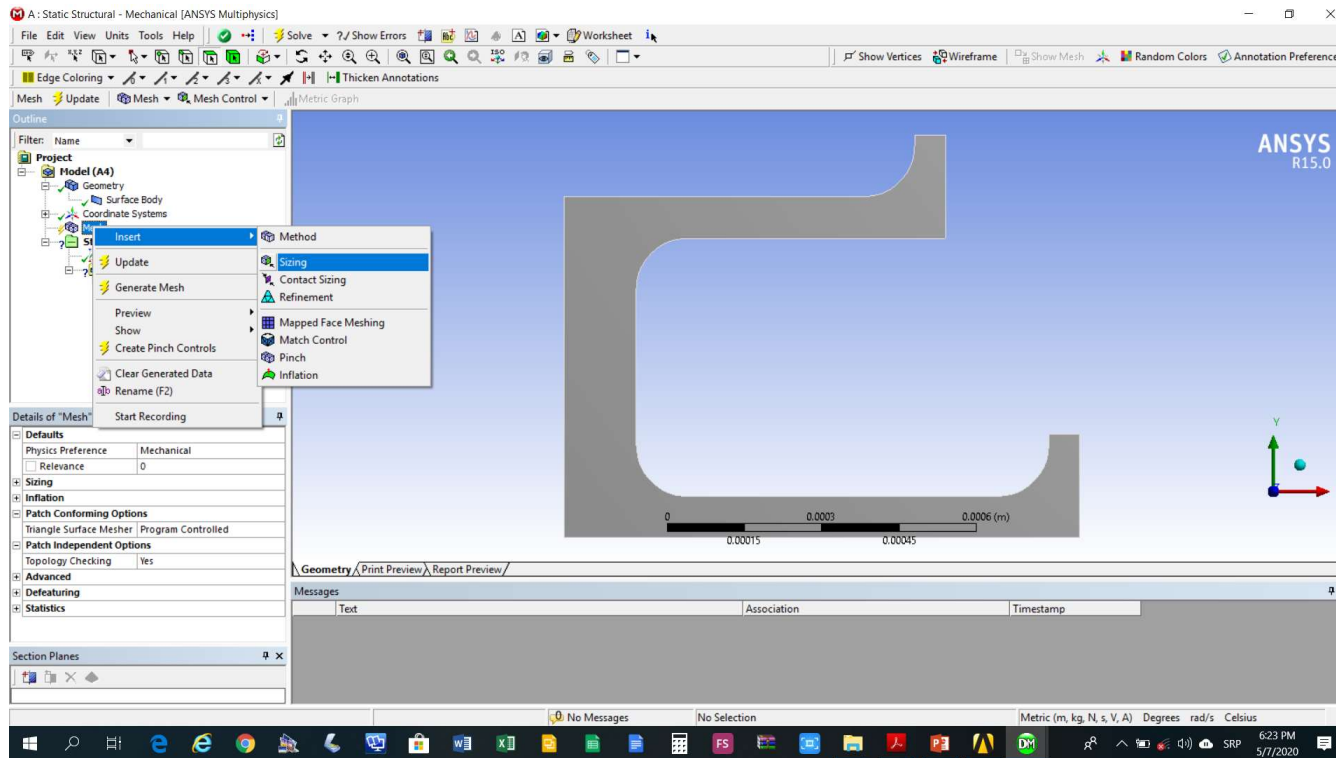
Modeliranje 2D problema

Dodjeliti odgovarajući materijal sa liste *Details of Surface Body->Material->Assignment* izabrati *Gray Cast Iron*



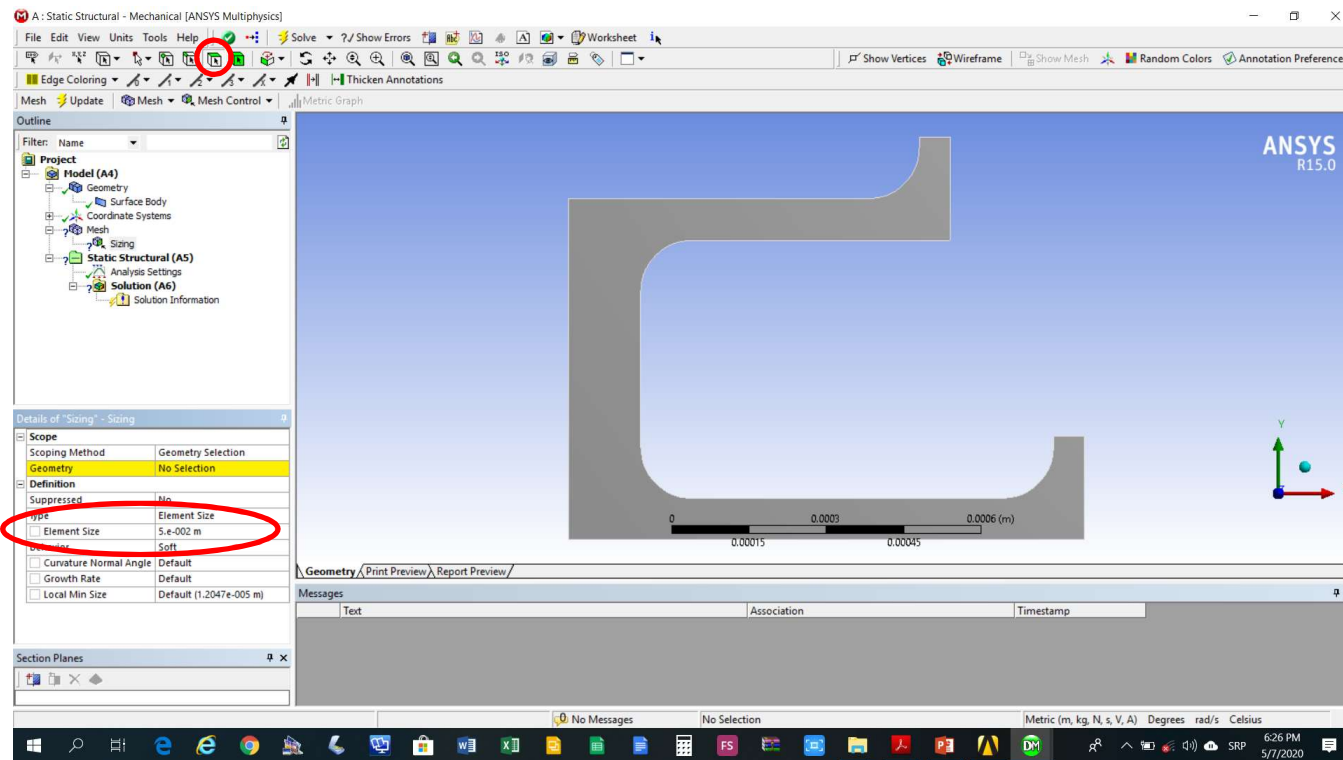
Modeliranje 2D problema

Definisati veličinu konačnih elemenata *Mesh*-
>*Insert*->*Sizing*



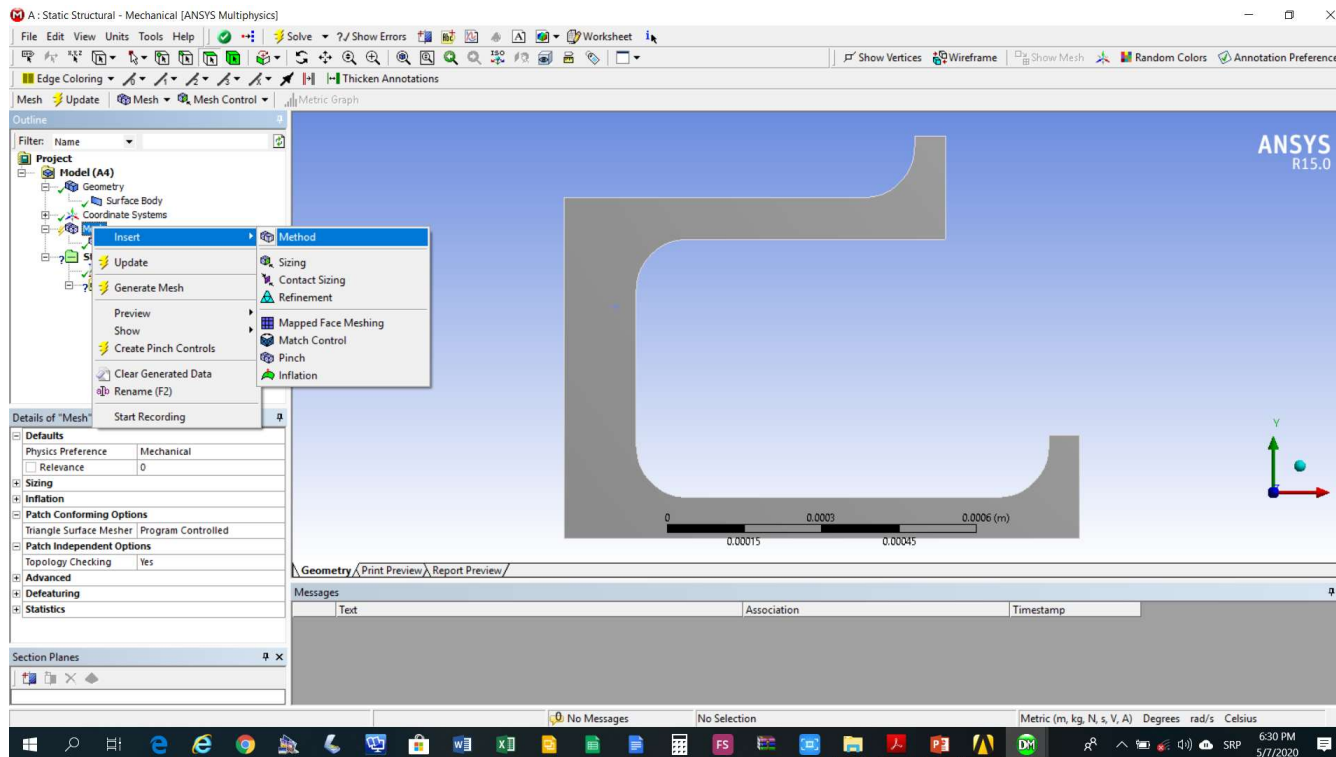
Modeliranje 2D problema

U polje *Details of Sizing*->*Element Size* unijeti 0.05 m. Izabrati površinu i aktivirati *Details of Sizing*->*Geometry*->*Apply*



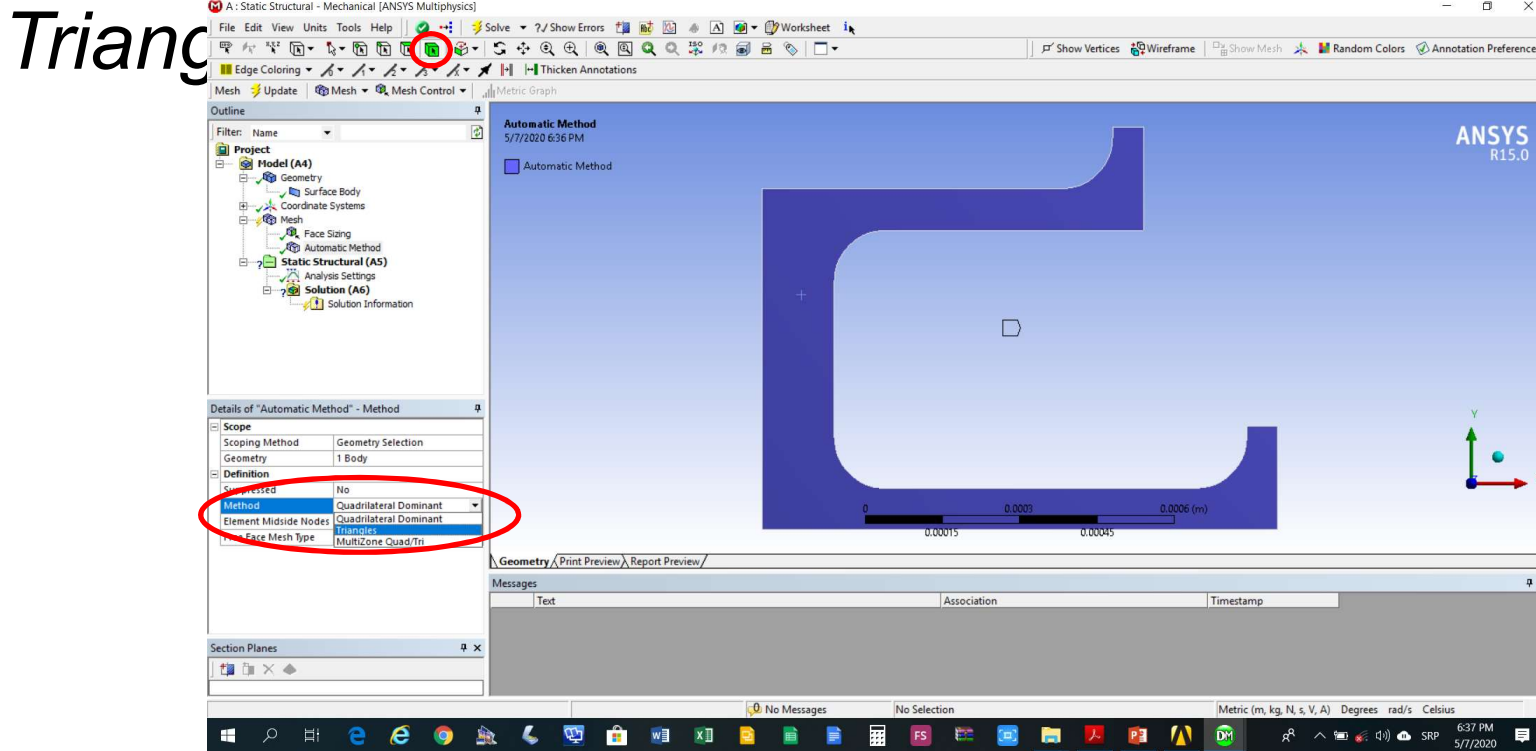
Modeliranje 2D problema

Definisati vrstu konačnih elemenata *Mesh*-
>*Insert*->*Method*



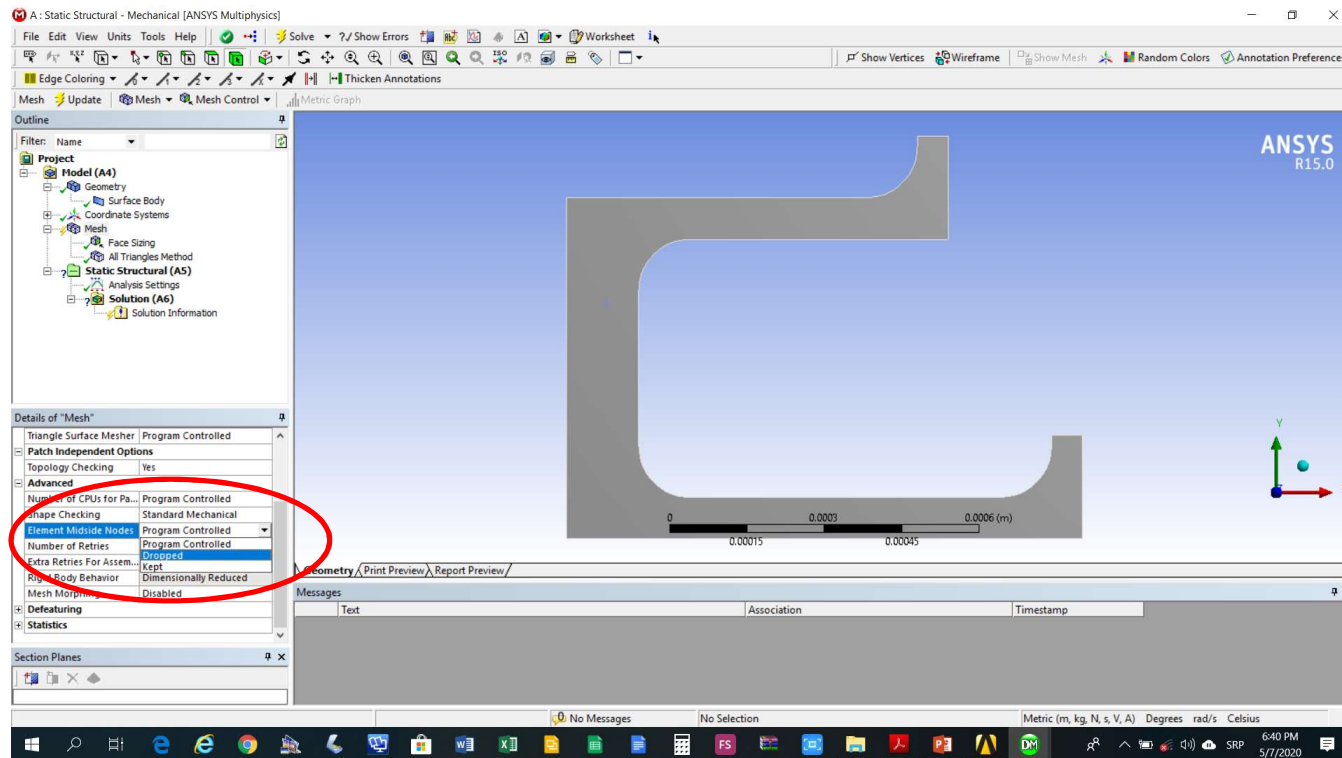
Modeliranje 2D problema

Izabrati tijelo i aktivirati *Details of Automatic Method*->*Geometry*->*Apply*. Iz liste *Details of Automatic Method*->*Method* izabrati opciju



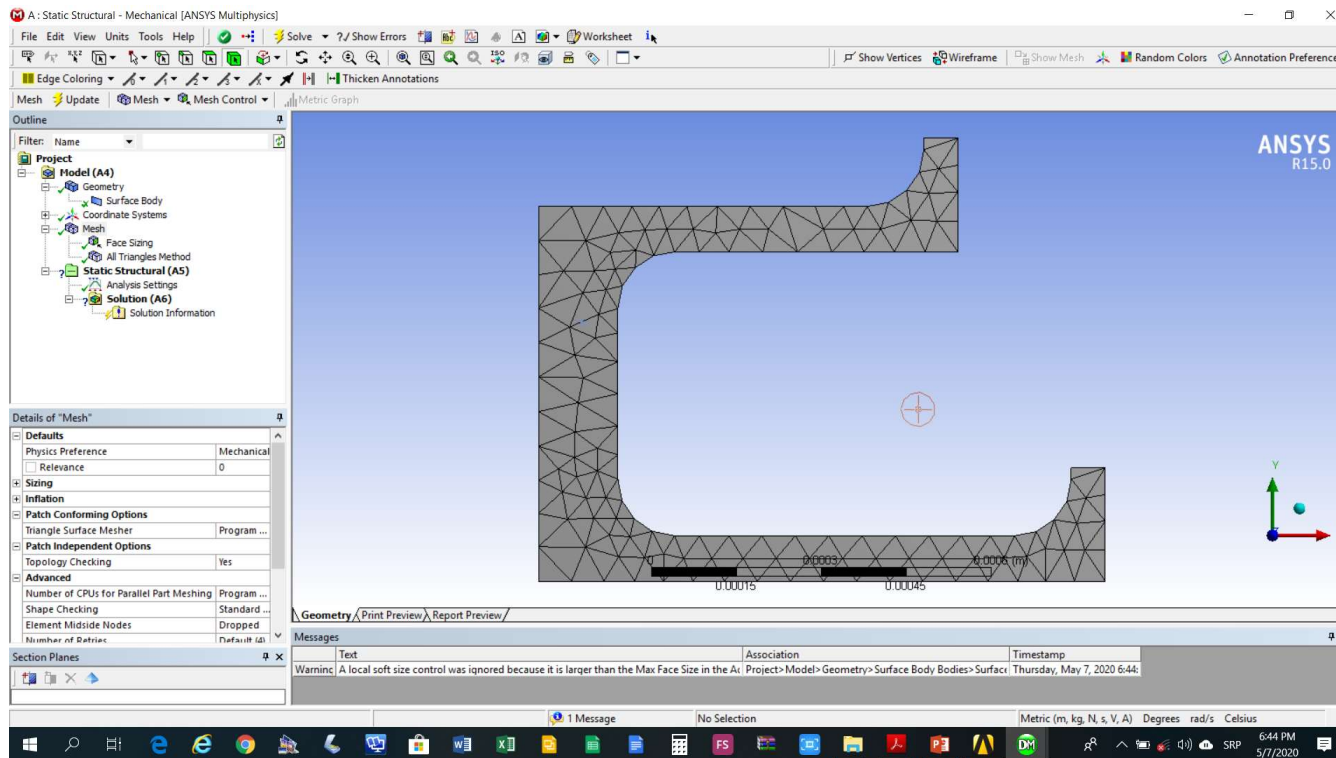
Modeliranje 2D problema

Iz liste *Details of Mesh*->*Advanced*->*Element Midside Nodes* izabrati opciju *Dropped* čime se vrši izbor linearnih trougaonih elemenata



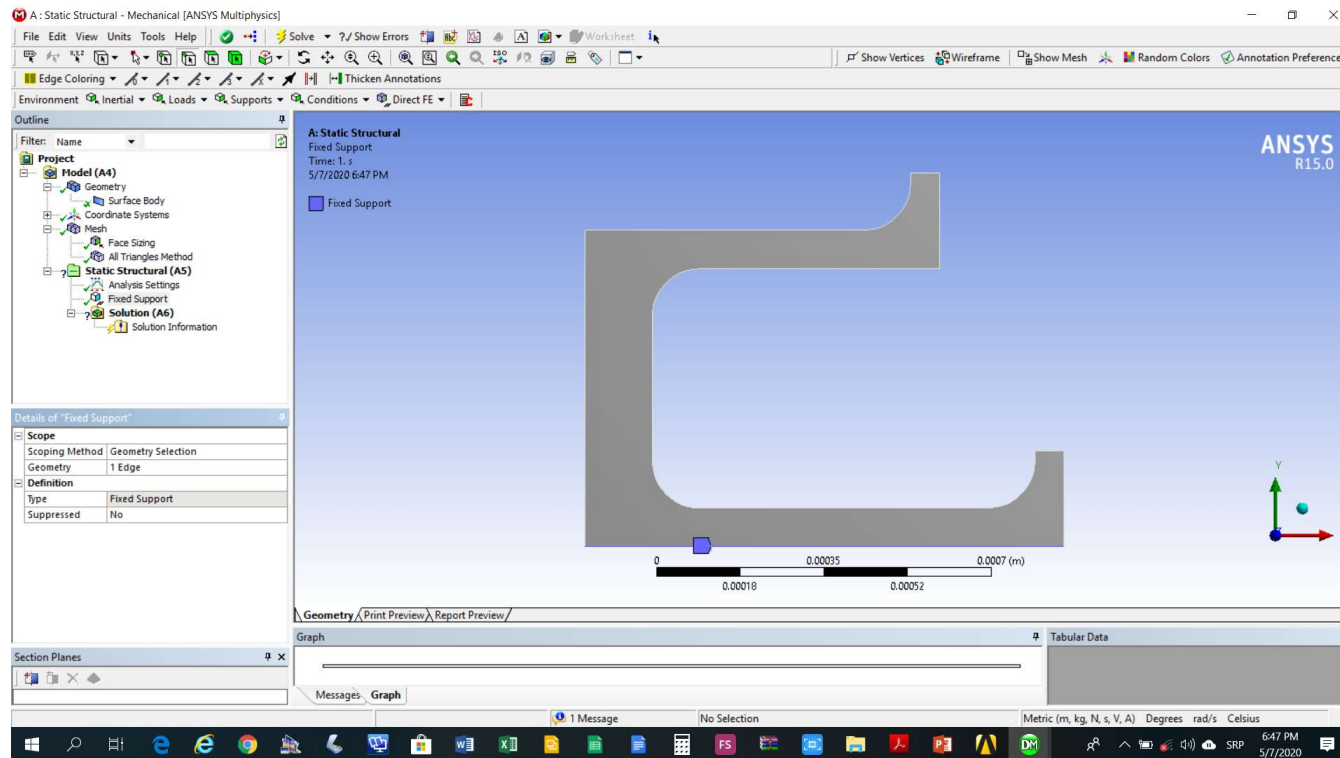
Modeliranje 2D problema

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



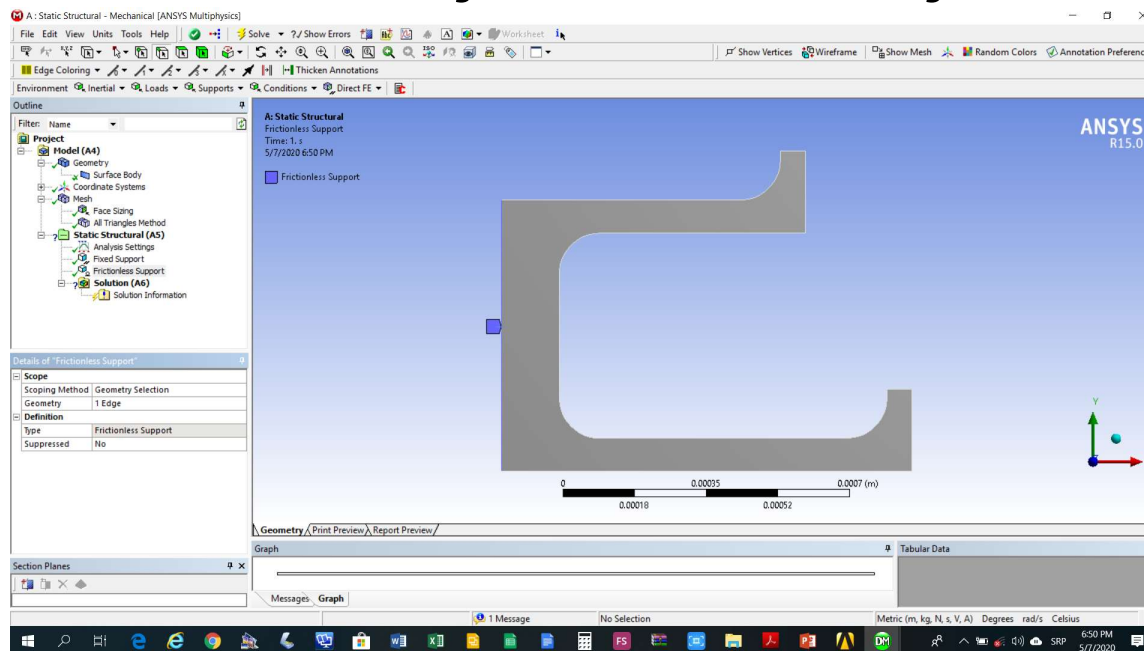
Modeliranje 2D problema

Dodati nepokretni oslonac na donju horizontalnu ivicu modela *Static Structural->Insert->Fixed Support*



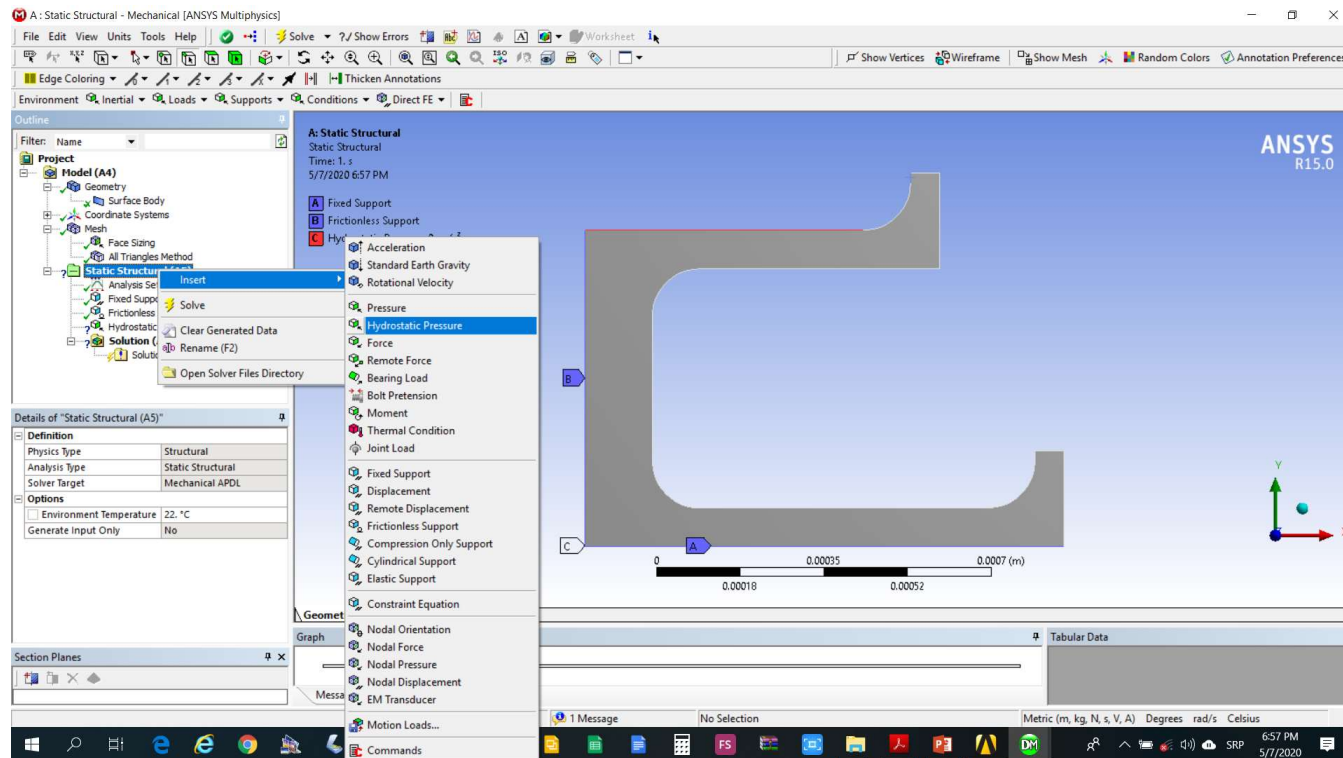
Modeliranje 2D problema

Dodati glatki oslonac na lijevu vertikalnu ivicu modela *Static Structural*->*Insert*->*Frictionless Support* što omogućava pomjeranja ili poprečno deformisanje ose simetrije



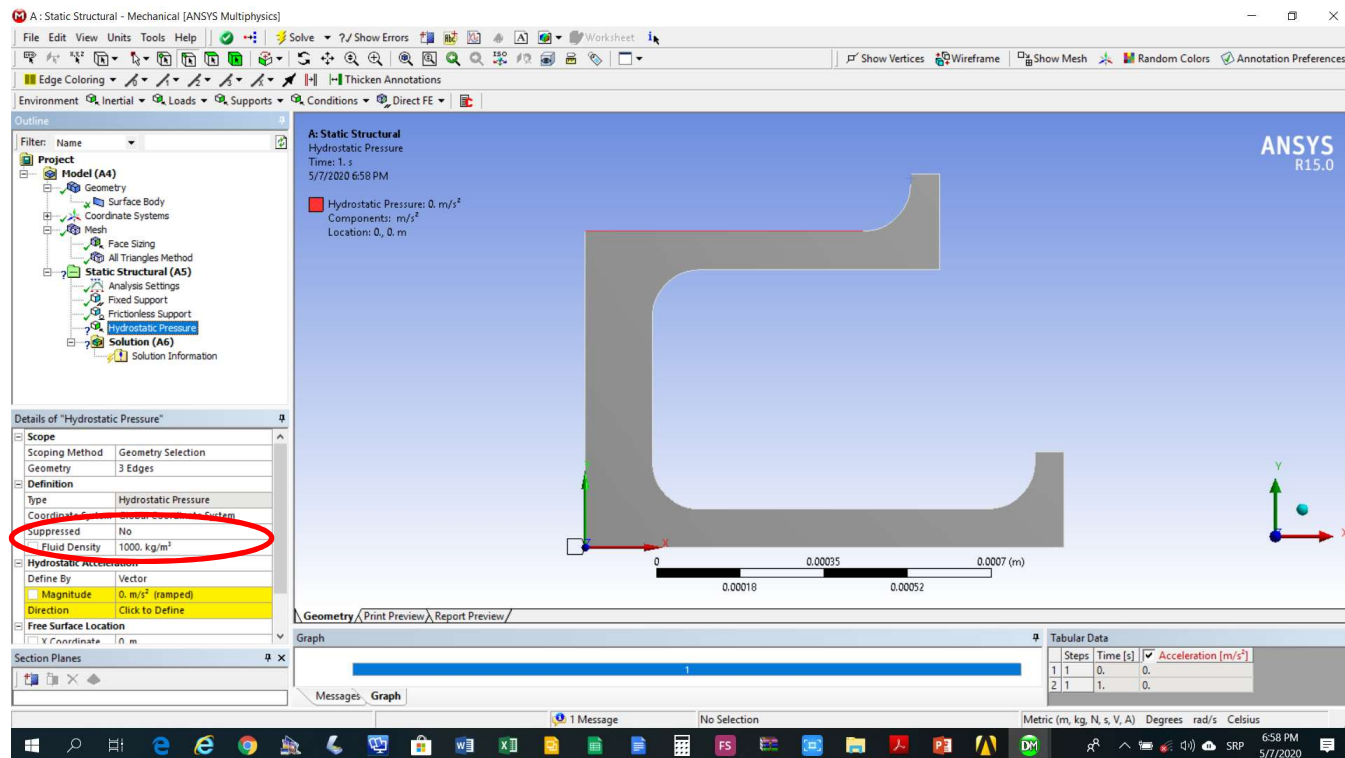
Modeliranje 2D problema

Dodati opterećenje uzrokovano težinom fluida
Static Structural->Insert->Hydrostatic Pressure



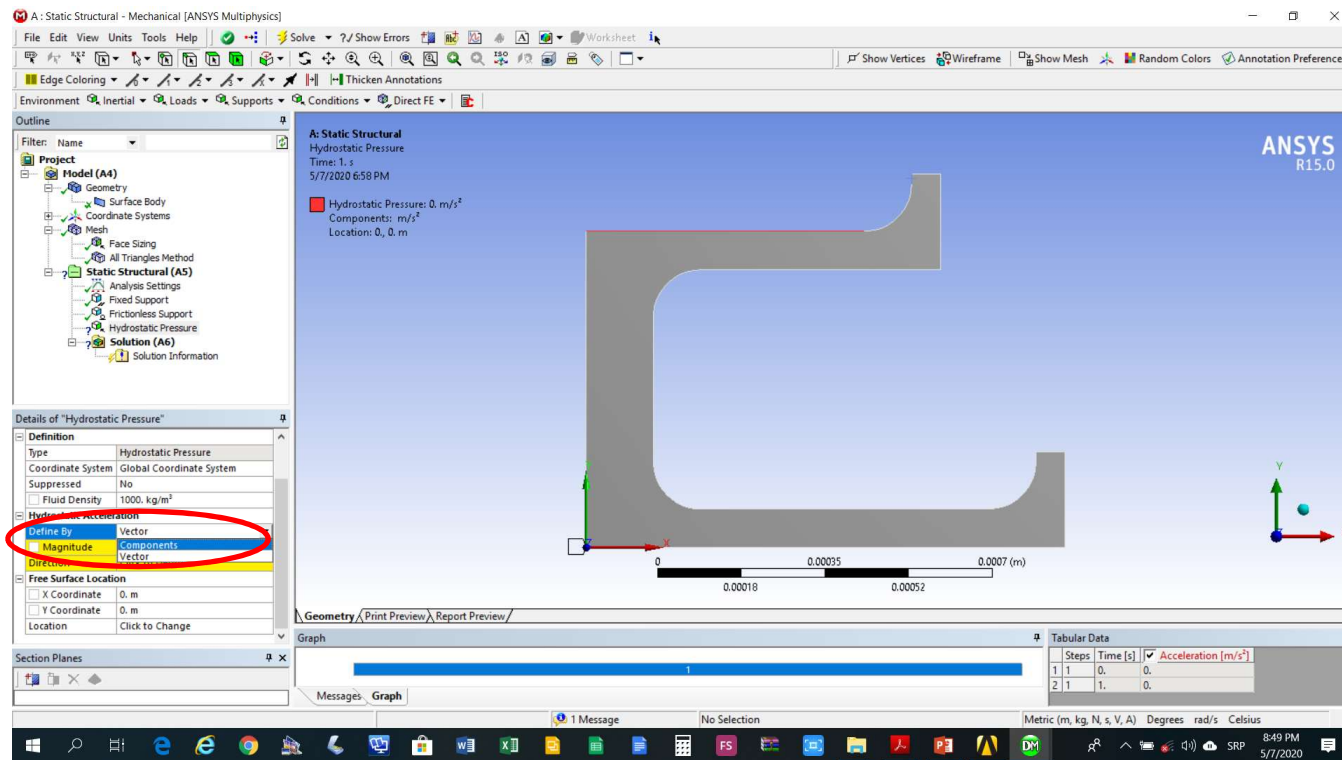
Modeliranje 2D problema

Definisati gustinu fluida unosom u polje *Details of Hydrostatic Pressure*->*Definition*->*Fluid Density*



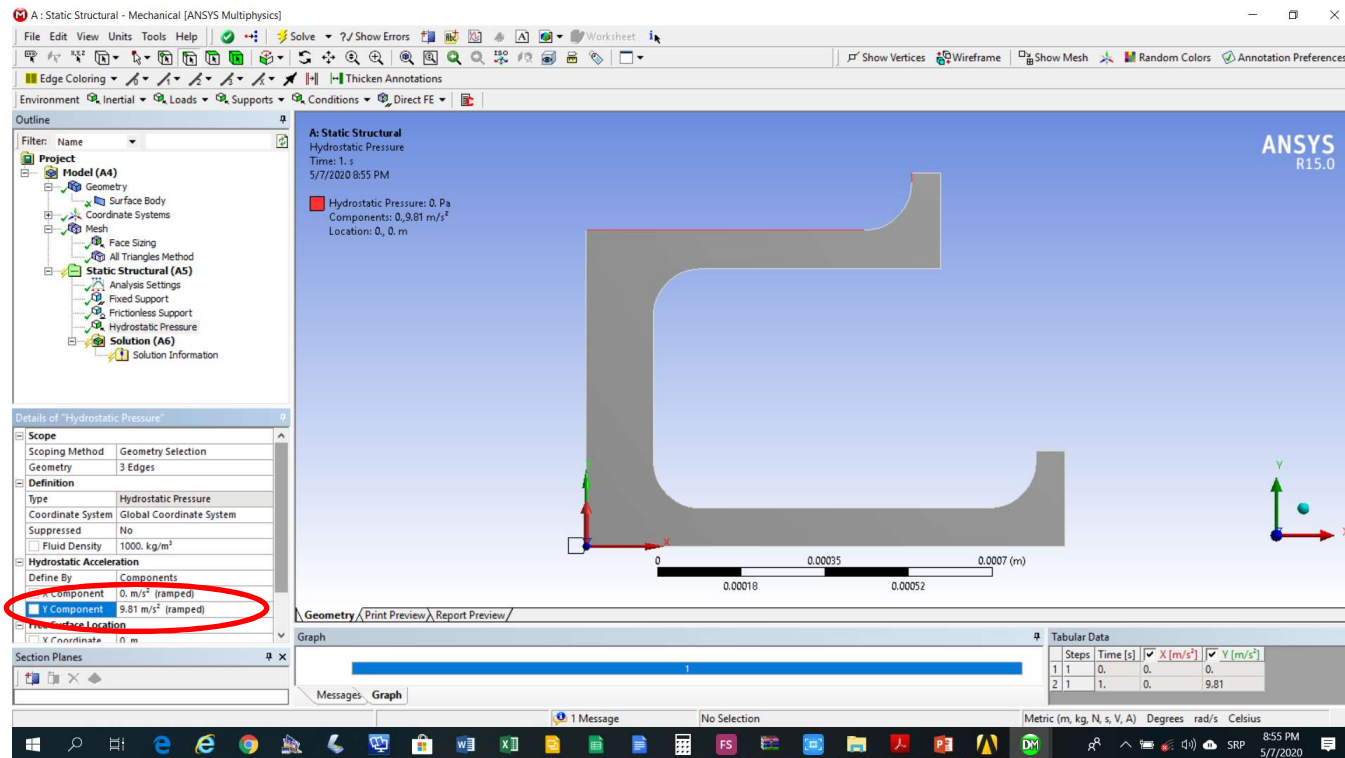
Modeliranje 2D problema

Sa liste *Details of Hydrostatic Pressure*->*Hydrostatic Acceleration*->*Define By* izabrati opciju *Components*



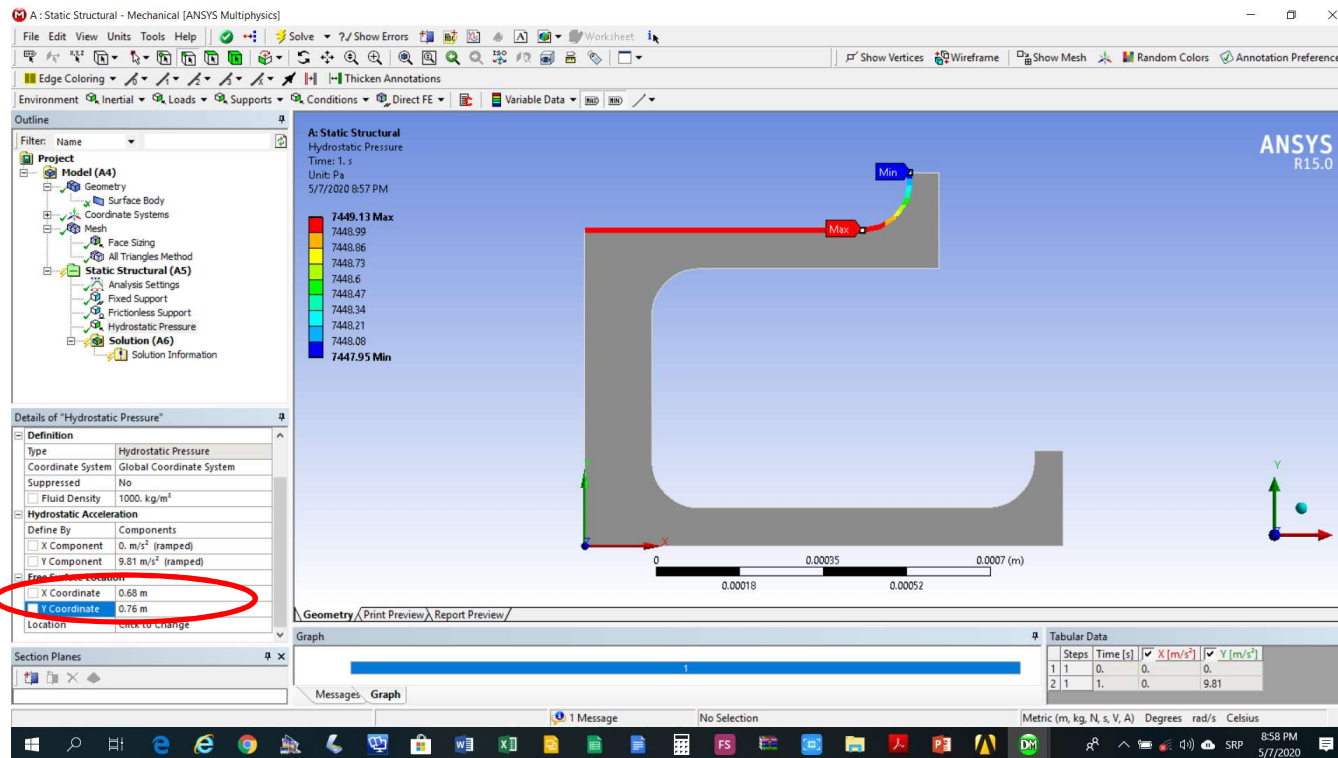
Modeliranje 2D problema

Definisati ubrzanje zemljine teže 9.81 m/s^2
unosom u polje *Hydrostatic Pressure*-
>*Hydrostatic Acceleration*->*Y Component*



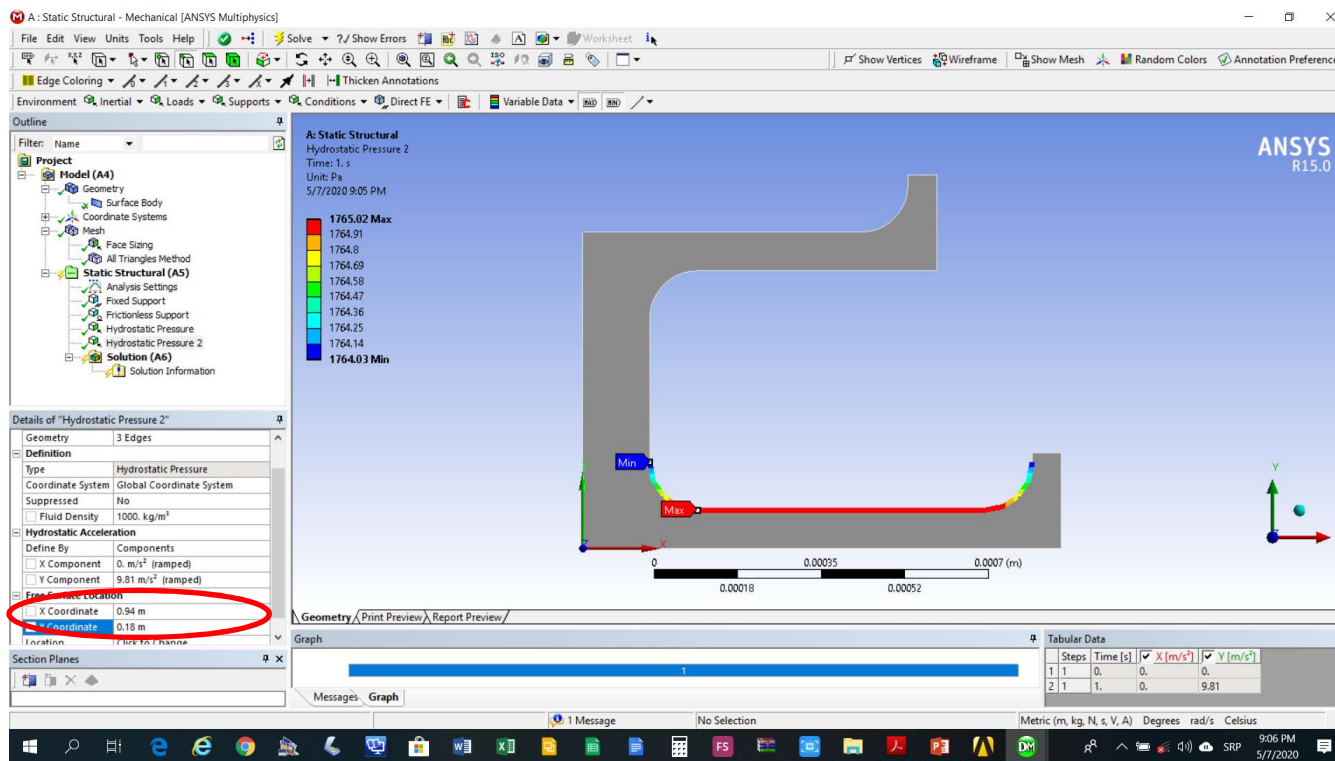
Modeliranje 2D problema

Unijeti u polja *Hydrostatic Pressure*->*Free Surface Location*->*X/Y Coordinate* 0.68m i 0.76m što odgovara položaju tačke na vrhu zaobljenja



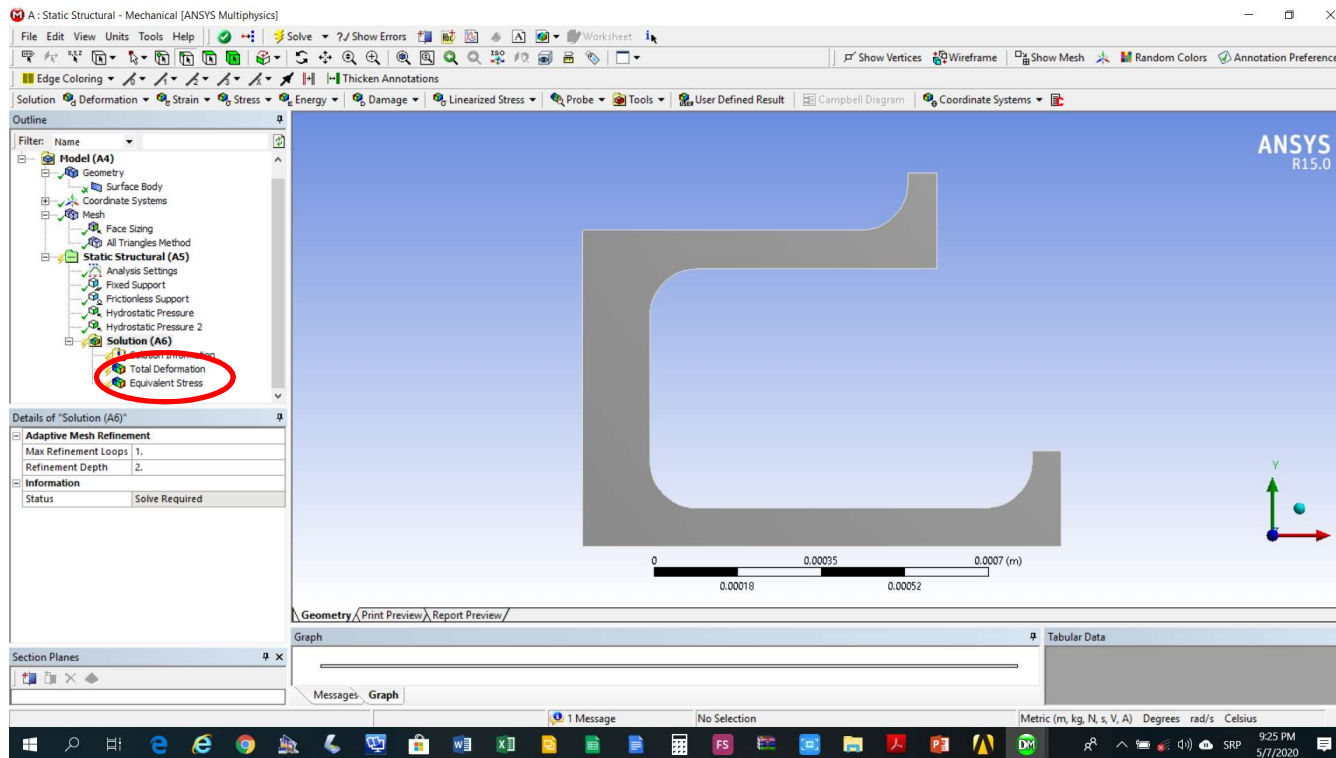
Modeliranje 2D problema

Unijeti u polja *Hydrostatic Pressure*->*Free Surface Location*->*X/Y Coordinate* 0.94m i 0.18m što odgovara položaju tačke na vrhu zaobljenja



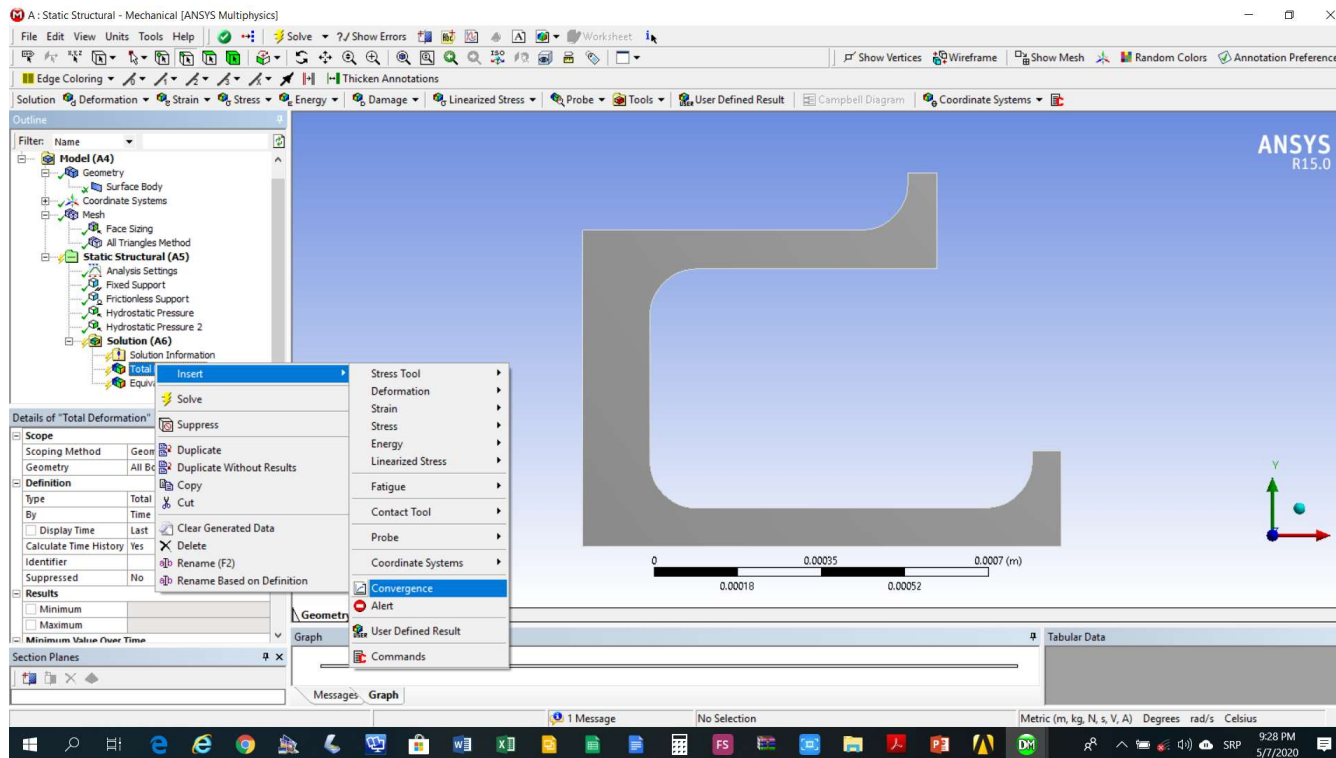
Modeliranje 2D problema

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



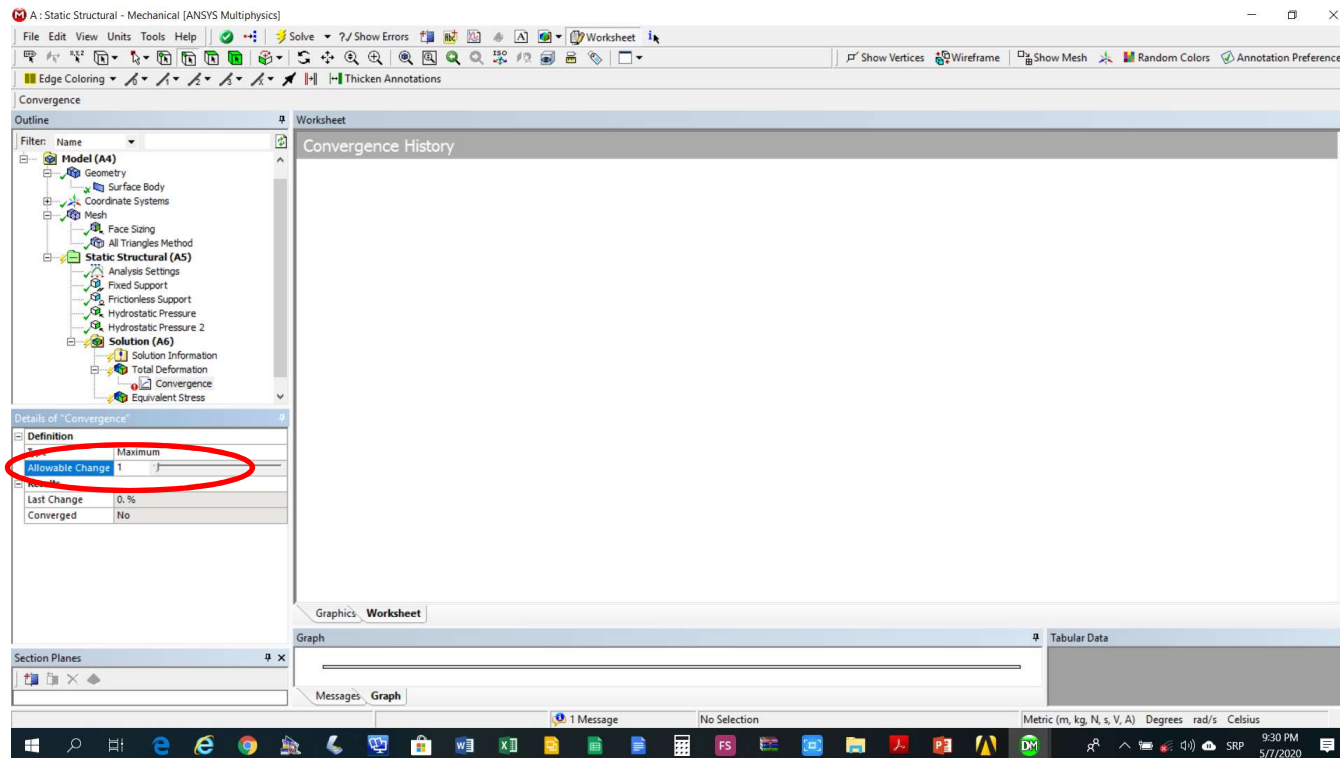
Modeliranje 2D problema

Umetnuti kontrolu konvergencije rješenja *Details of Total Deformation*->*Insert*->*Convergence*



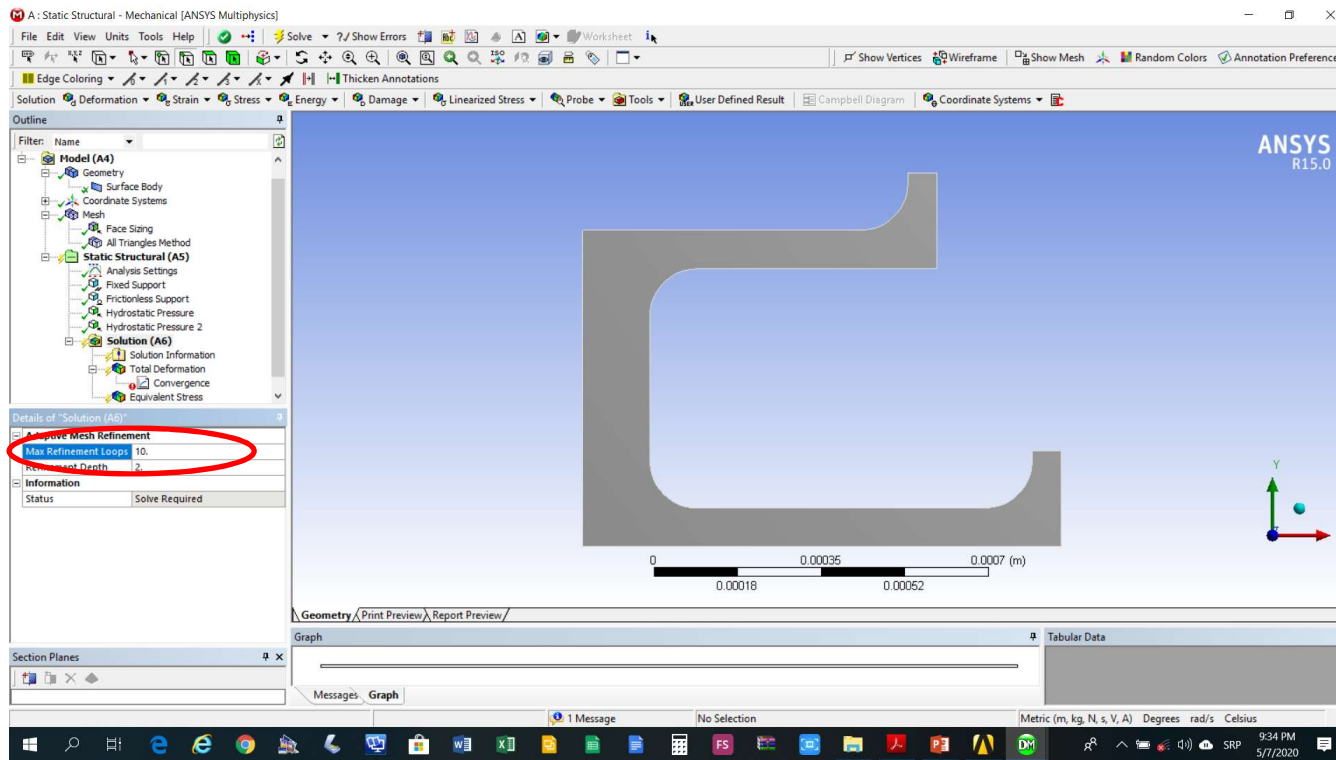
Modeliranje 2D problema

Definisati ciljnju tačnost konvergencije rješenja 1% u polje *Details of Convergence*->*Definition*->*Allowable Change*



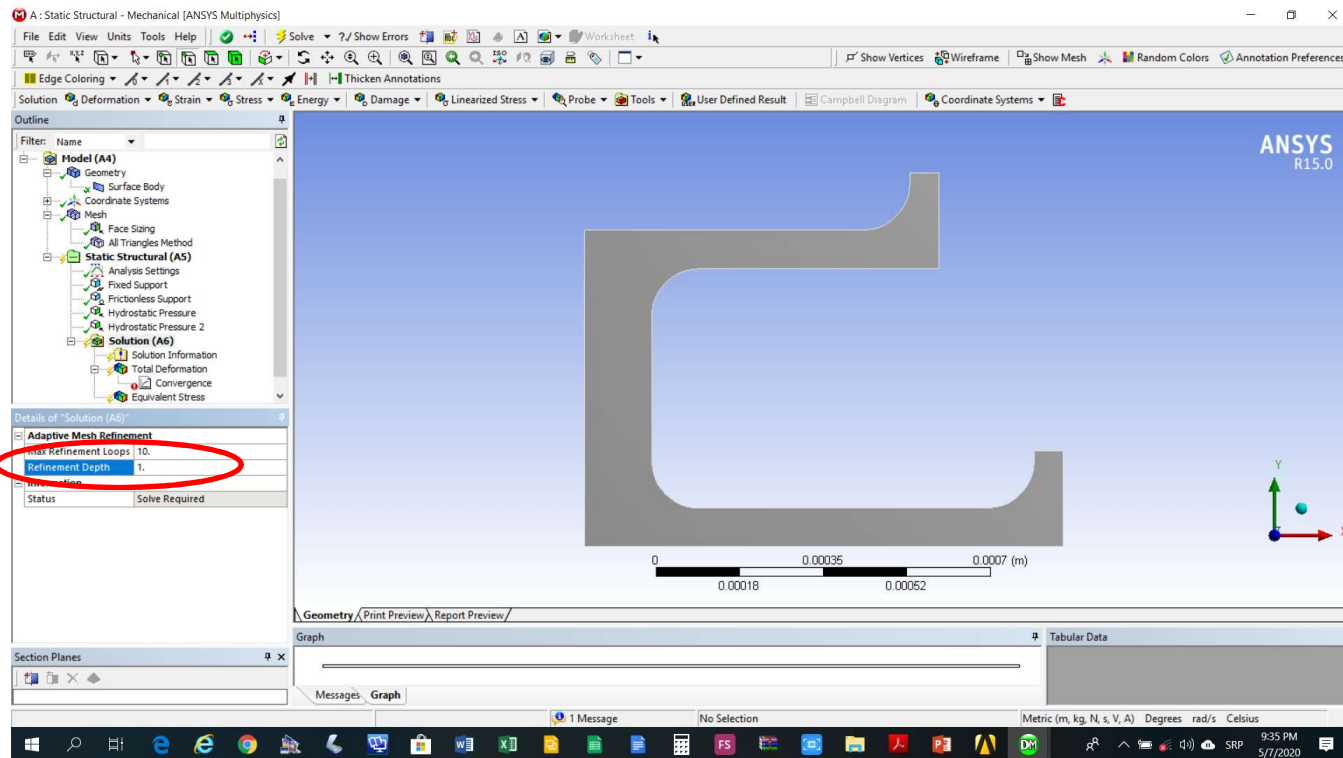
Modeliranje 2D problema

Definisati broj adaptacija mreže na 10 u polje
*Details of Solution->Adaptive Mesh Refinement-
>Max Refinement Loops*



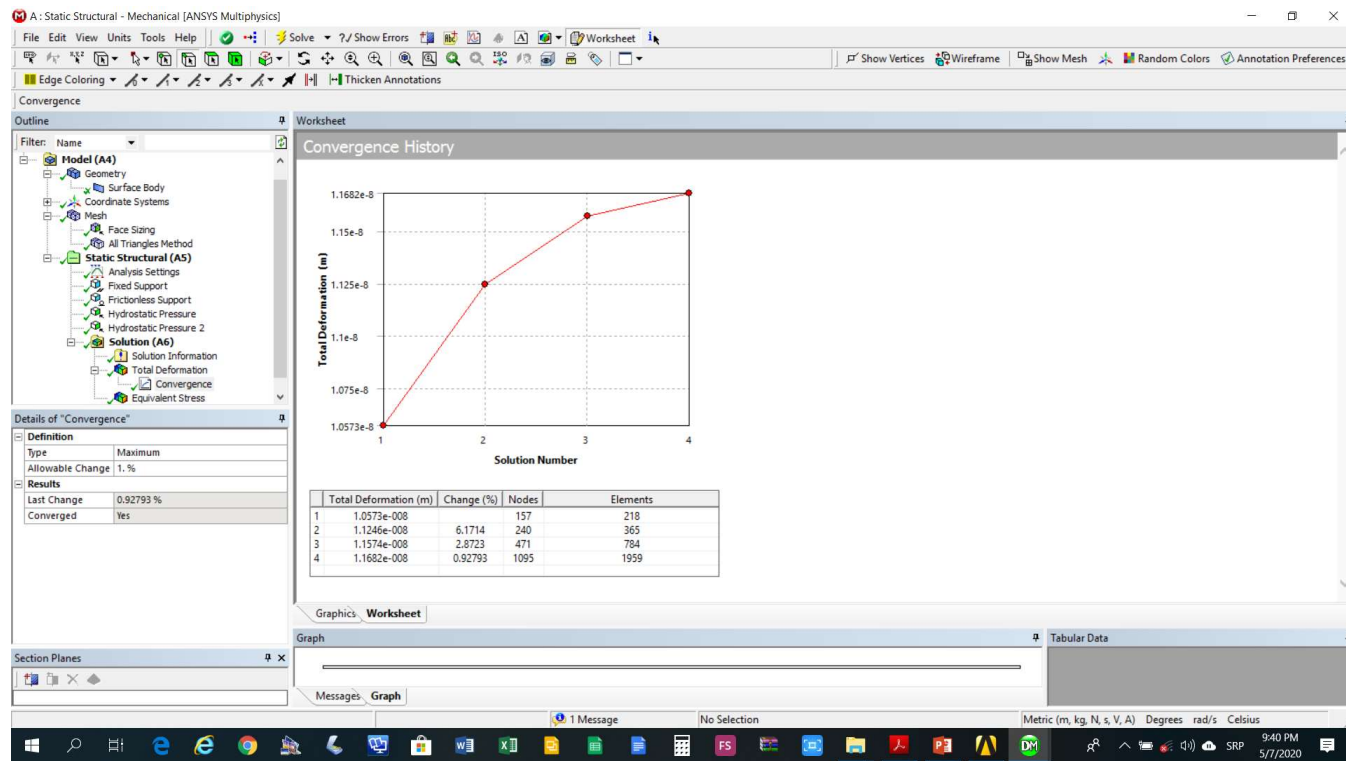
Modeliranje 2D problema

Definisati intezitet adaptacija mreže na 1 u polje
Details of Solution->*Adaptive Mesh Refinement*-
>*Refinement Depth* (moguće od 0 do 3)



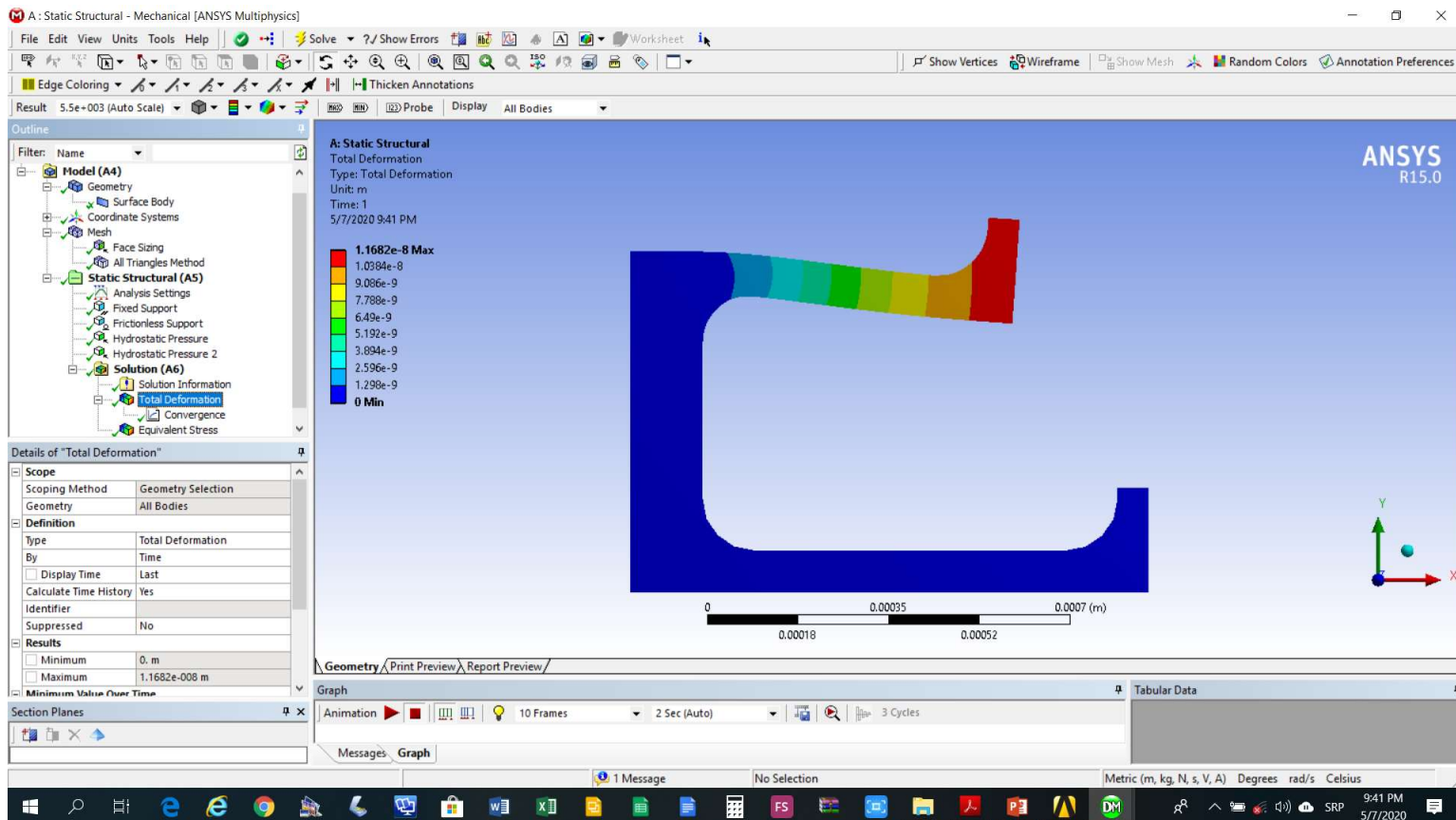
Modeliranje 2D problema

Riješiti problem. Riješenje se traži sve dok razlika između dvije uzastopne iteracije ne bude manja od 1% ili bude realizovano 10 adaptacija mreže



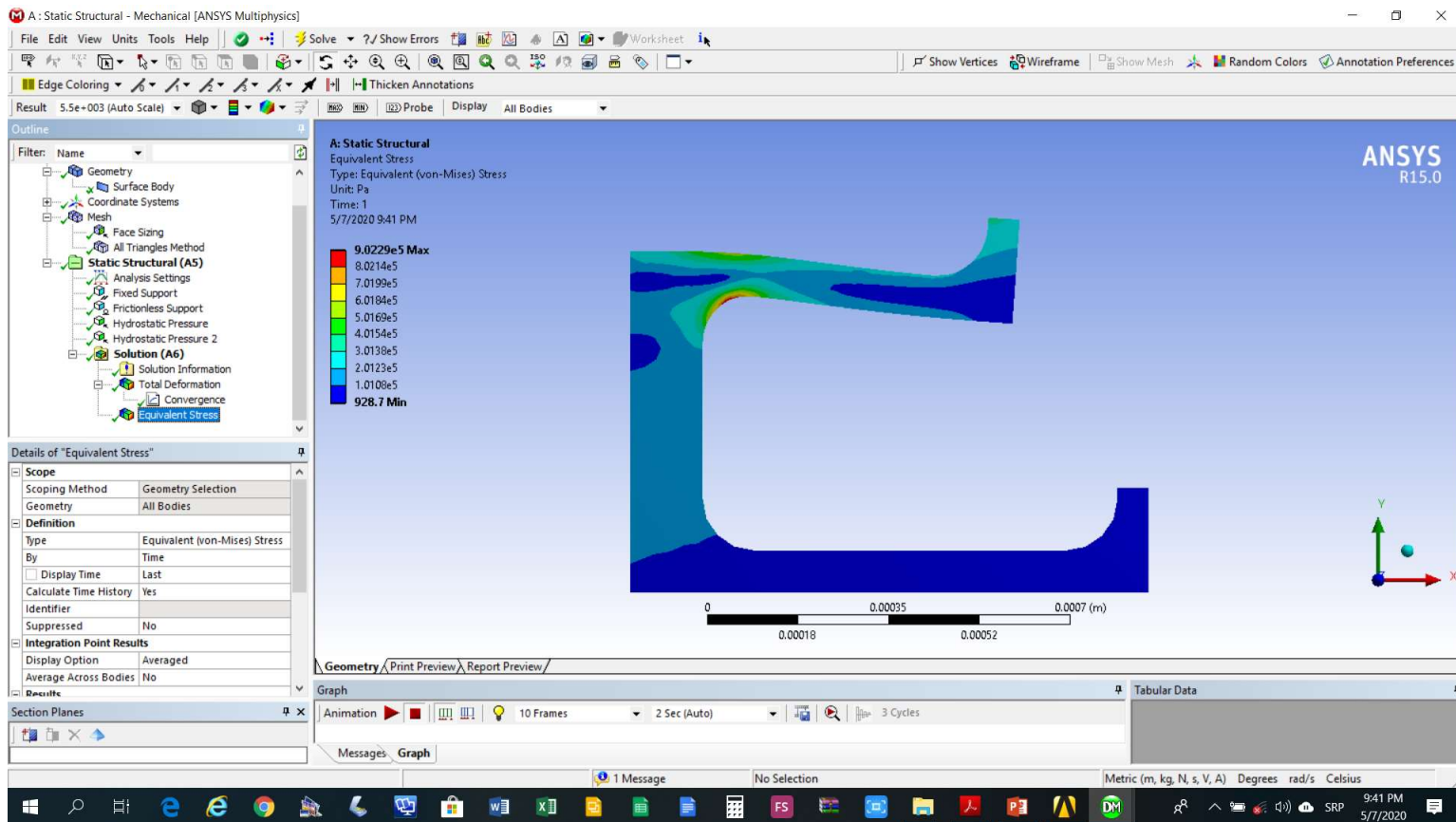
Modeliranje 2D problema

Polje pomjeranja



Modeliranje 2D problema

Polje ekvivalentnih napona



Modeliranje 2D problema

Dodati opterećenje od sopstvene težine fontane
Static Structural->Insert->Standard Earth Gravity

