Kielce University Of Technology

Computer Aided Engineering Works

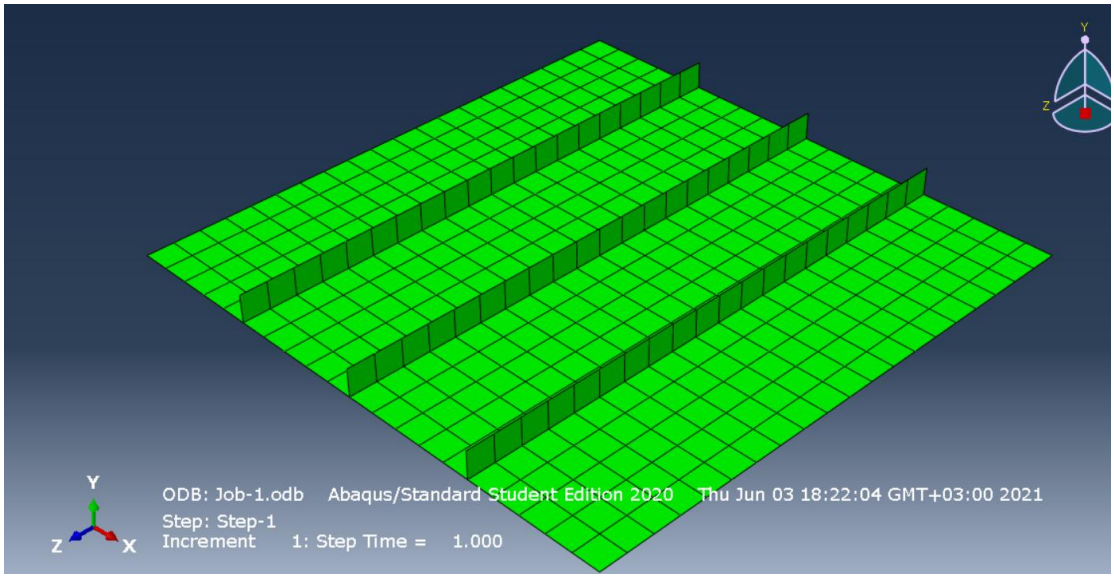
EDA KAŞ
402264

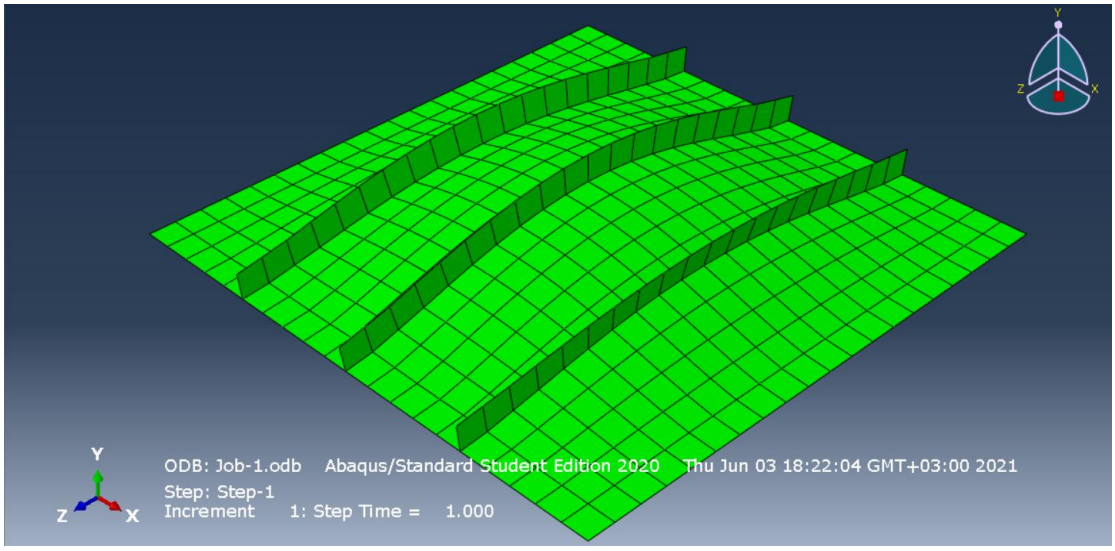
My Project topic is:

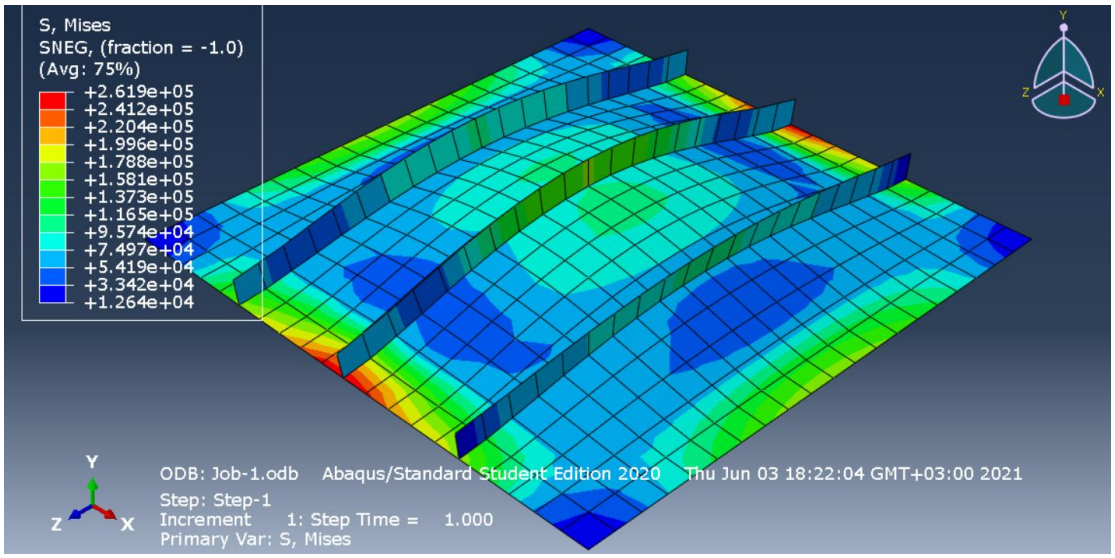
10.5 Example: blast loading on a stiffened plate

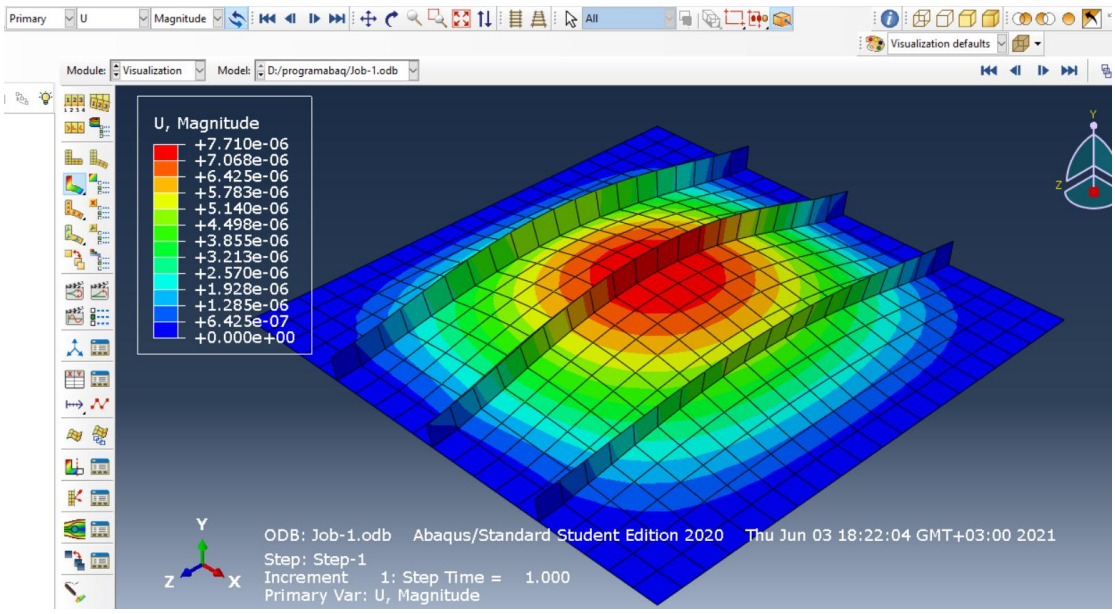
*I created a three-dimensional deformable part featuring an extruded shell base to represent the plate. I used about 5.0 piece size and named the piece Plate.
*I defined the stiffener geometry, added three vertical lines extending up from the plate.
*I constrained the three vertical lines so they are of equal length, and dimension one of them so that it is 0.1 m long.
*I splited the plate at the points where it intersects the stiffeners.
*I dimensioned the horizontal distance between the plate endpoints, and set the value to 2.0 m.
*I applied equal length constraints to the four horizontal segments of the line.
*I extruded the sketch to a depth of 2.0 m to create the plate.
*I defined the material and section properties for the plate and the stiffeners.
*I created a material named "Steel" with a "mass density" of 7800 kg/m3, a "Young's modulus" of 210.0E9 Pa, and a "Poisson's" ratio of 0.3.
*The initial yield stress is 300 MPa, and the yield stress increases to 400 MPa at a plastic strain of 35%.
* The data are regularized by Abaqus/Explicit by expanding the data to 15 equally spaced points with increments of 0.025.
*I was set the regularization tolerance back to the default value (0.03) and remove the additional pair of data points.
*I created two homogeneous shell section properties, each referring to the steel material definition but specifying different shell thicknesses.
*I named the first shell section property PlateSection, select Steel as the material, and specify 0.025 m as the value for the Shell thickness.
*I named the second shell section property StiffSection, select Steel as the material, and specify 0.0125 m as the value for the Shell thickness.
*I assigned the StiffSection definition to the stiffeners (I used [Shift] + Click to select multiple regions in the viewport).
*I was query the shell normals (ToolsQuery) and note the color of the side of the plate facing the stiffeners (brown is the positive side; purple is the negative side).
*I was assign the PlateSection definition to the regions of the plate.
*I was in the Edit Section Assignment dialog box, set the shell offset to Top surfaced if the brown (positive) side of the plate faces the stiffeners and Bottom surfaced if the purple (negative) side faces the stiffeners.
*I was to verify the offset, selected ViewPart Display Options.
* I was in the Part Display Options dialog box that appears, toggled on Render shell thickness. I modified the offset to removed any overlap.
*I created one set named Edge for the plate edges and one set named Center at the center of the intersection of the plate and the middle stiffener.I was To create the set Center, you need to first partition the edge of the original part in half using the Partition Edge: I Entered Parameter tool.
*I created a single dynamic, explicit step. I named the step Blast.I applied blast loading. I entered a value of 50E-3 s for the time period of the step.

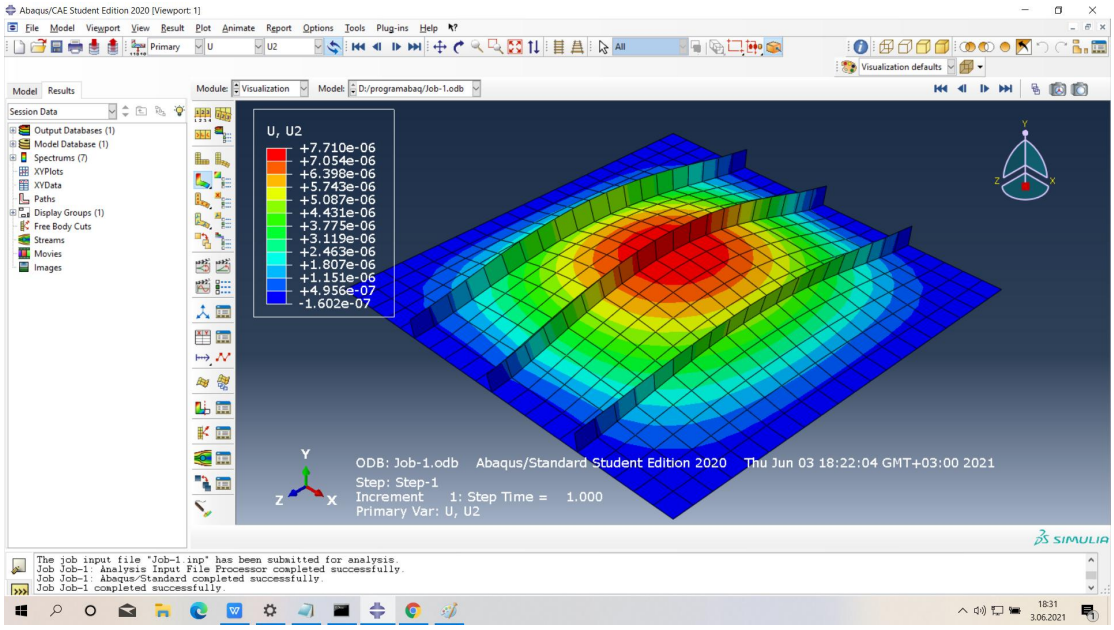
*I tried to limit the number of frames written during the analysis to keep the size of the output database file reasonable.
*In this analysis saving information every 2 ms should provide sufficient detail to study the response of the structure.
*I created a history output request named Center-U2 for the step Blast.
*I selected Center as the output domain, and select U2 as the translation output variable.
*I entered 500 as the number of intervals at which the output will be saved during the analysis.
*In the step Blast, I created a Symmetry/Antisymmetry/Encastre mechanical boundary condition named Fix edges.
*I applied the boundary condition to the edges of the plate using the geometry set Edge, and specify ENCASTRE (U1 = U2 = U3 = UR1 = UR2 = UR3 = 0) to fully constrain the set.
*In the Model Tree, I double-clicked the Loads container.
*In the Created Load dialog box that appears, I named the load Pressure load and selected Blast as the step in which it will be applied.
*I selected Mechanical as the load category and Pressure as the load type.
*I selected all the surfaces associated with the plate. When I the appropriate surfaces are selected, I clicked Done.
*I created a job named BlastLoad. I blasted load on a flat plate with stiffeners: S4R elements (20x20 mesh) Normal stiffeners (20x2).
*I saved your model in a model database file, and submit the job for analysis.

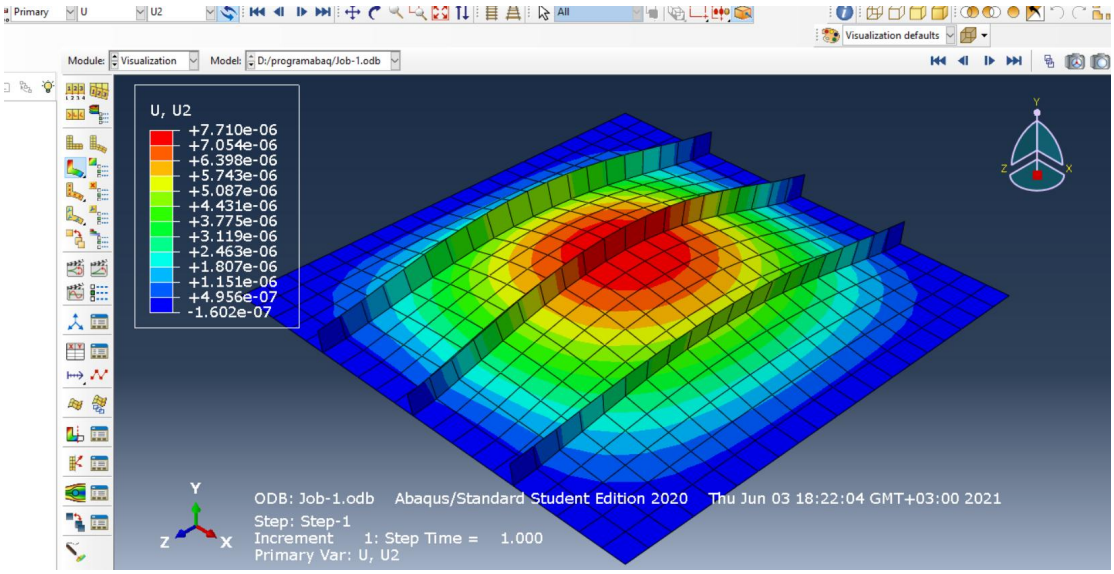


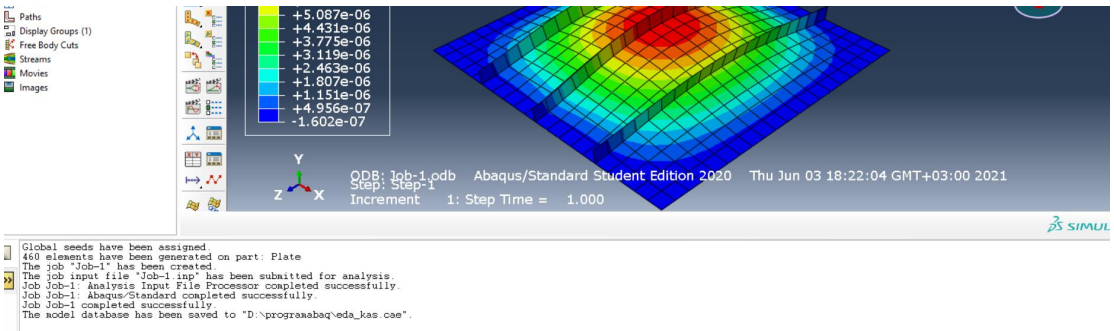

ODB: Job-1.odb Abaqus/Standard Student Edition 2020 Thu Jun 03 18:22:04 GMT+03:00 2021
Step: Step-1
Increment 1: Step Time = 1.000


S, Mises
SNEG, (fraction = -1.0)
(Avg: 75%)
+2.619e+05
+2.412e+05
+2.204e+05
+1.996e+05
+1.788e+05
+1.581e+05
+1.373e+05
+1.165e+05
+9.574e+04
+7.497e+04
+5.419e+04
+3.342e+04
+1.264e+04
ODB: Job-1.odb Abaqus/Standard Student Edition 2020 Thu Jun 03 18:22:04 GMT+03:00 2021
Step: Step-1
Increment 1: Step Time = 1.000
Primary Var: S, Mises

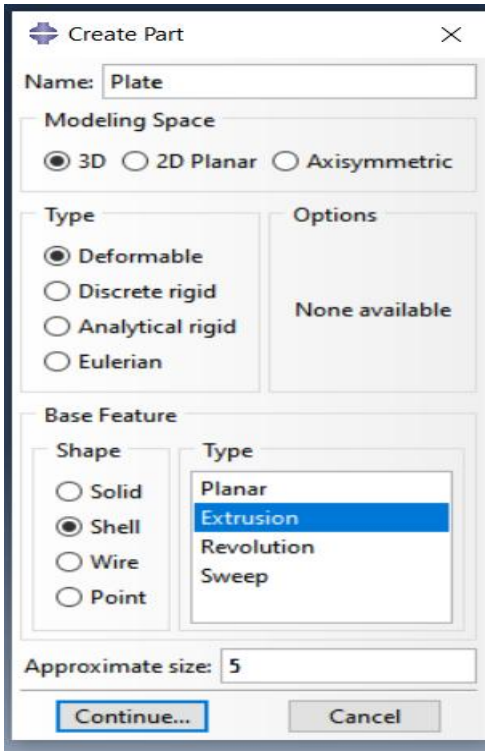

Primary
U
Magnitude
Visualization defaults
Module: Visualization
Model: D:/programabaq/Job-1.odb
U, Magnitude
+7.710e-06
+7.068e-06
+6.425e-06
+5.783e-06
+5.140e-06
+4.498e-06
+3.855e-06
+3.213e-06
+2.570e-06
+1.928e-06
+1.285e-06
+6.425e-07
+0.000e+00
ODB: Job-1.odb Abaqus/Standard Student Edition 2020 Thu Jun 03 18:22:04 GMT+03:00 2021
Step: Step-1
Increment 1: Step Time = 1.000
Primary Var: U, Magnitude


Abaqus/CAE Student Edition 2020 [Viewport: 1]
File Model Viewport View Result Plot Animate Report Options Tools Plug-ins Help
Model Results
Session Data
Output Databases (1)
Model Database (1)
Spectrums (7)
XYPlots
XYData
Paths
Display Groups (1)
Free Body Cuts
Streams
Movies
Images
Module: Visualization
Model: D:/programabaq/Job-1.odb
U, U2
+7.710e-06
+7.054e-06
+6.398e-06
+5.743e-06
+5.087e-06
+4.431e-06
+3.775e-06
+3.119e-06
+2.463e-06
+1.807e-06
+1.151e-06
+4.956e-07
-1.602e-07
ODB: Job-1.odb Abaqus/Standard Student Edition 2020 Thu Jun 03 18:22:04 GMT+03:00 2021
Step: Step-1
Increment 1: Step Time = 1.000
Primary Var: U, U2
The job input file "Job-1.inp" has been submitted for analysis.
Job Job-1: Analysis Input File Processor completed successfully.
Job Job-1: Abaqus/Standard completed successfully.
Job Job-1 completed successfully.
18:31
3.06.2021


Primary U U2
Visualization defaults
Module: Visualization
Model: D:/programabaq/Job-1.odb
U, U2
+7.710e-06
+7.054e-06
+6.398e-06
+5.743e-06
+5.087e-06
+4.431e-06
+3.775e-06
+3.119e-06
+2.463e-06
+1.807e-06
+1.151e-06
+4.956e-07
-1.602e-07
ODB: Job-1.odb Abaqus/Standard Student Edition 2020 Thu Jun 03 18:22:04 GMT+03:00 2021
Step: Step-1
Increment 1: Step Time = 1.000
Primary Var: U, U2

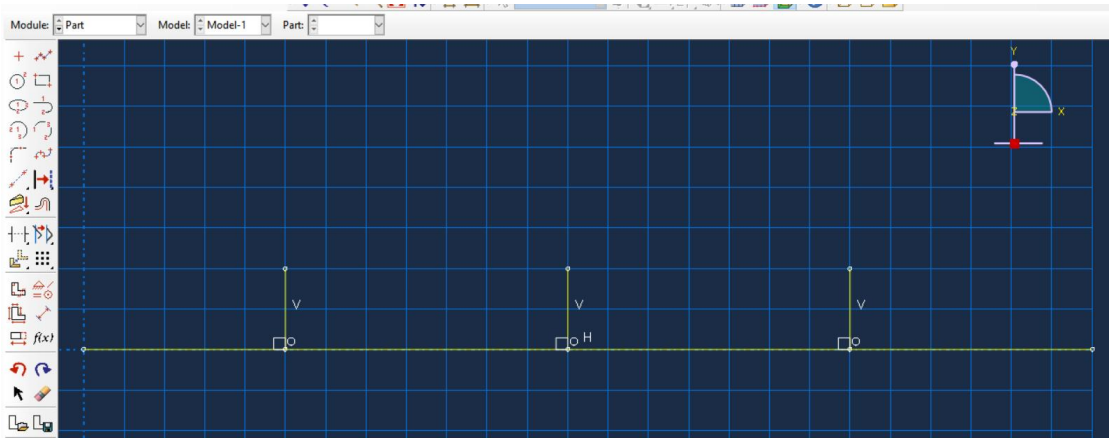

Paths
Display Groups (1)
Free Body Cuts
Streams
Movies
Images
+5.087e-06
+4.431e-06
+3.775e-06
+3.119e-06
+2.463e-06
+1.807e-06
+1.151e-06
+4.956e-07
-1.602e-07
ODB: Job-1.odb Abaqus/Standard Student Edition 2020 Thu Jun 03 18:22:04 GMT+03:00 2021
Step: Step-1
Increment 1: Step Time = 1.000
Global seeds have been assigned.
460 elements have been generated on part: Plate
The job "Job-1" has been created.
The job input file "Job-1.inp" has been submitted for analysis.
Job Job-1: Analysis Input File Processor completed successfully.
Job Job-1: Abaqus/Standard completed successfully.
Job Job-1 completed successfully.
The model database has been saved to "D:\programabaq\eda_kas.cae".

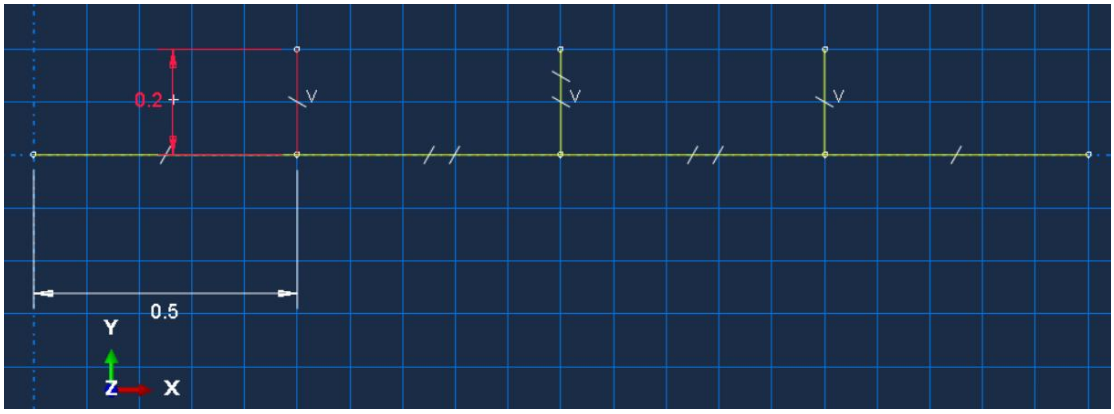
Firstly, click on “Create Part”. I choose what I have to choose.



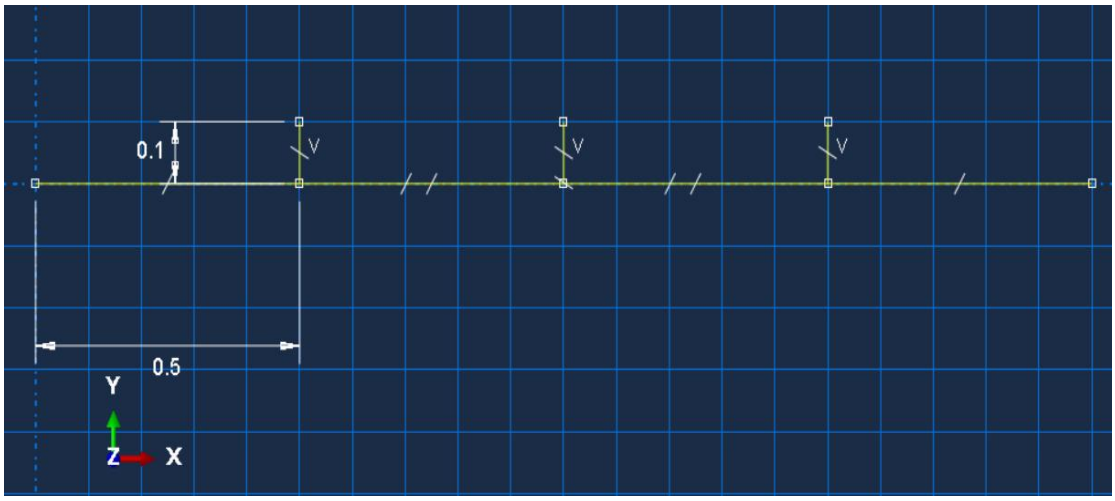
Some of my transactions:

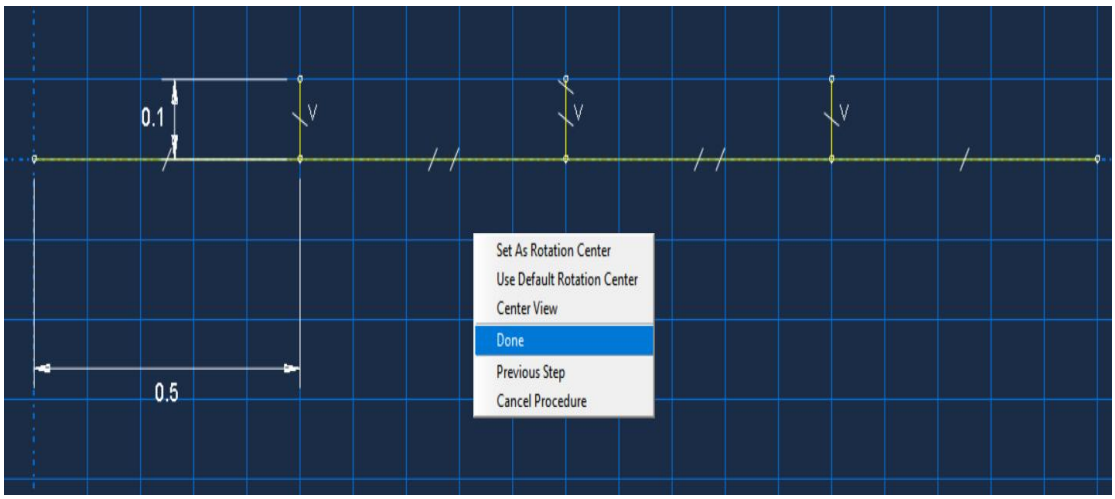
I click on" Create Line " and show the direction I will draw.

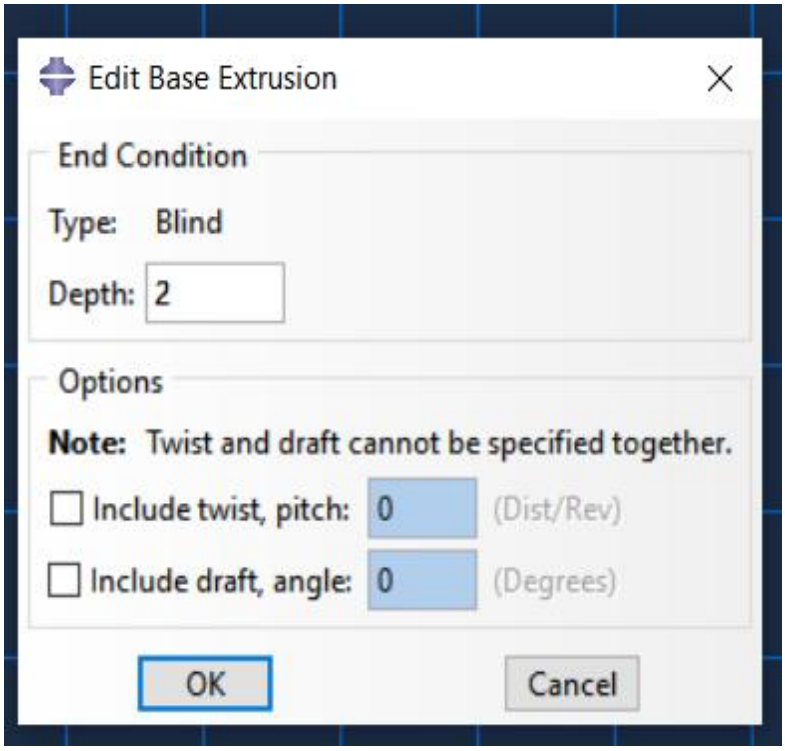


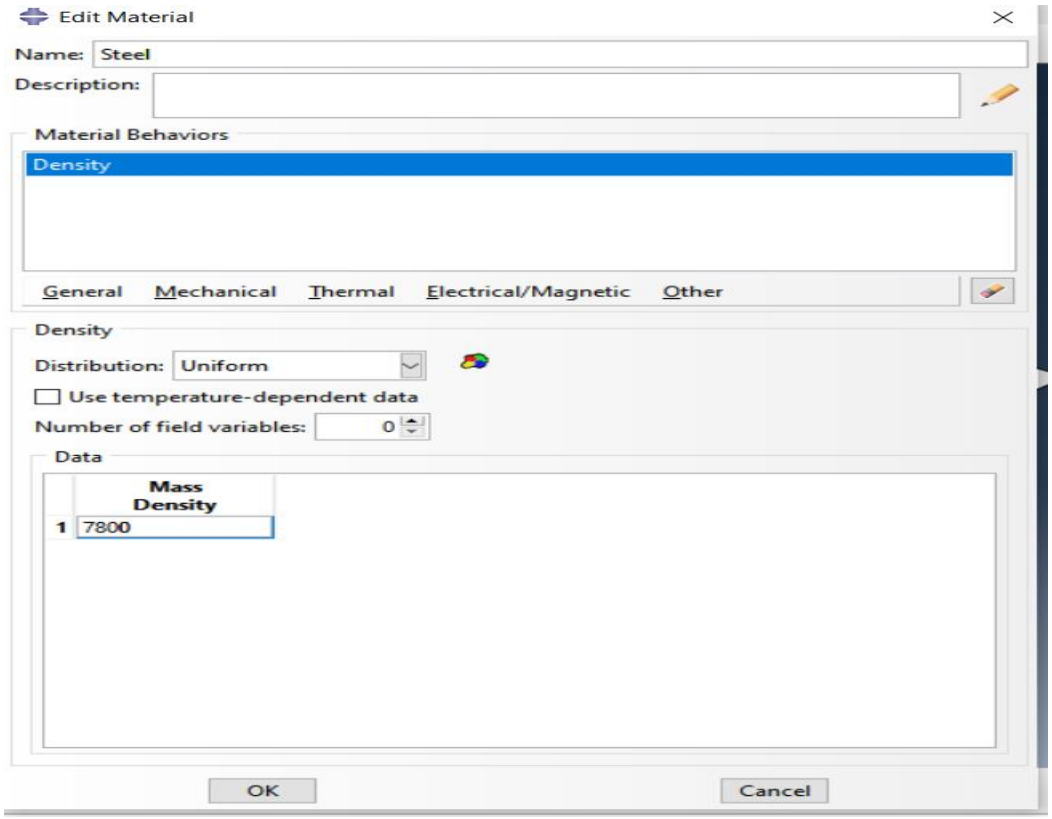


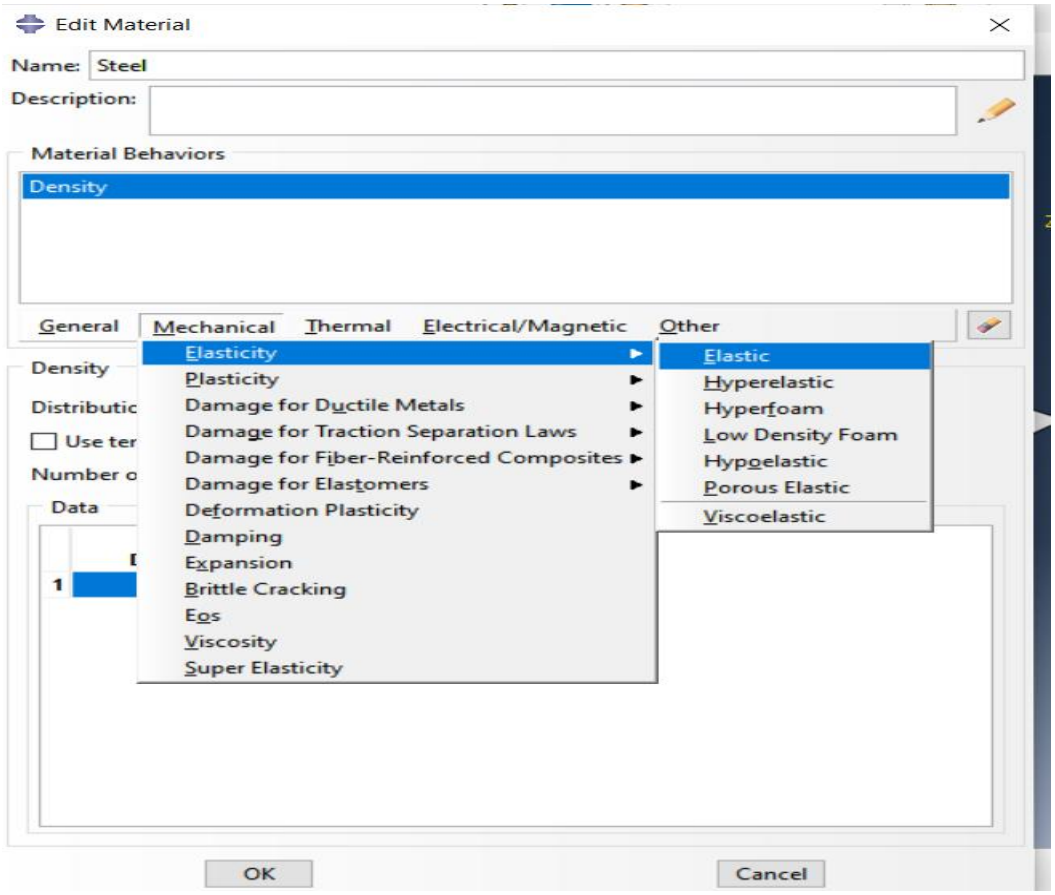
We click "Add Constraint" and choose "equal Length" from it

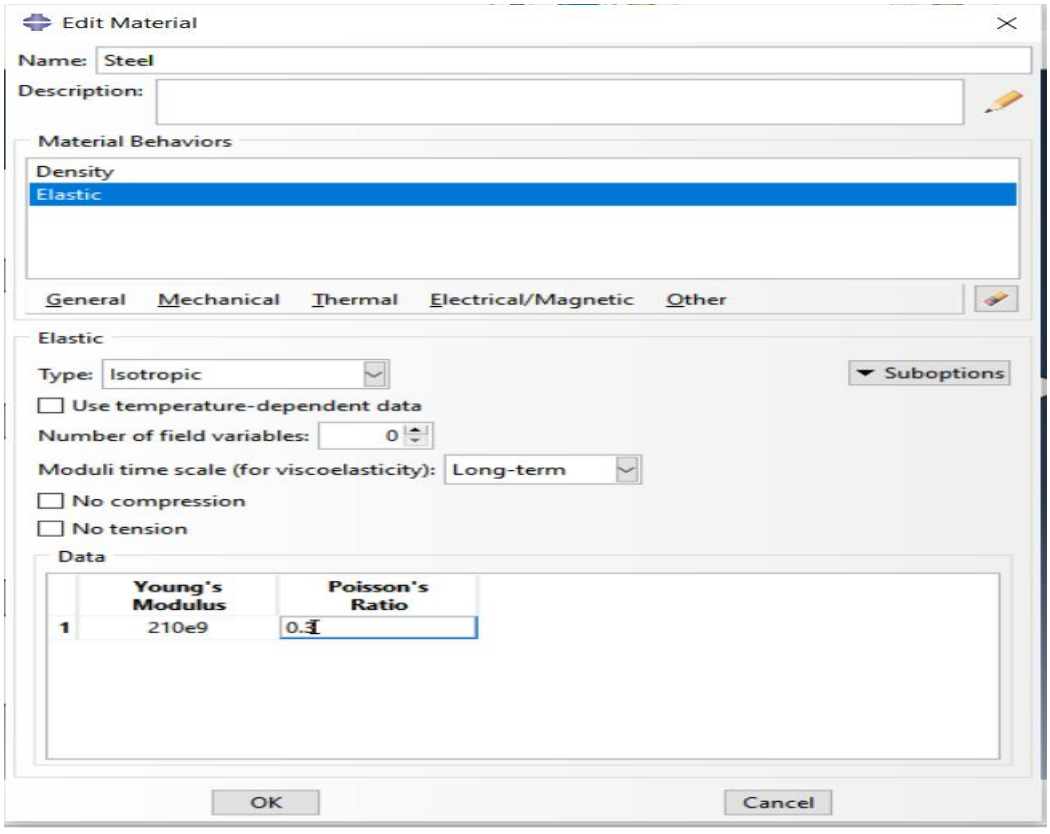


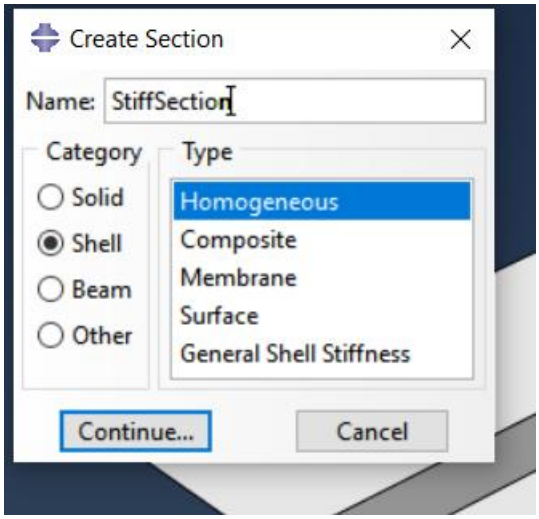


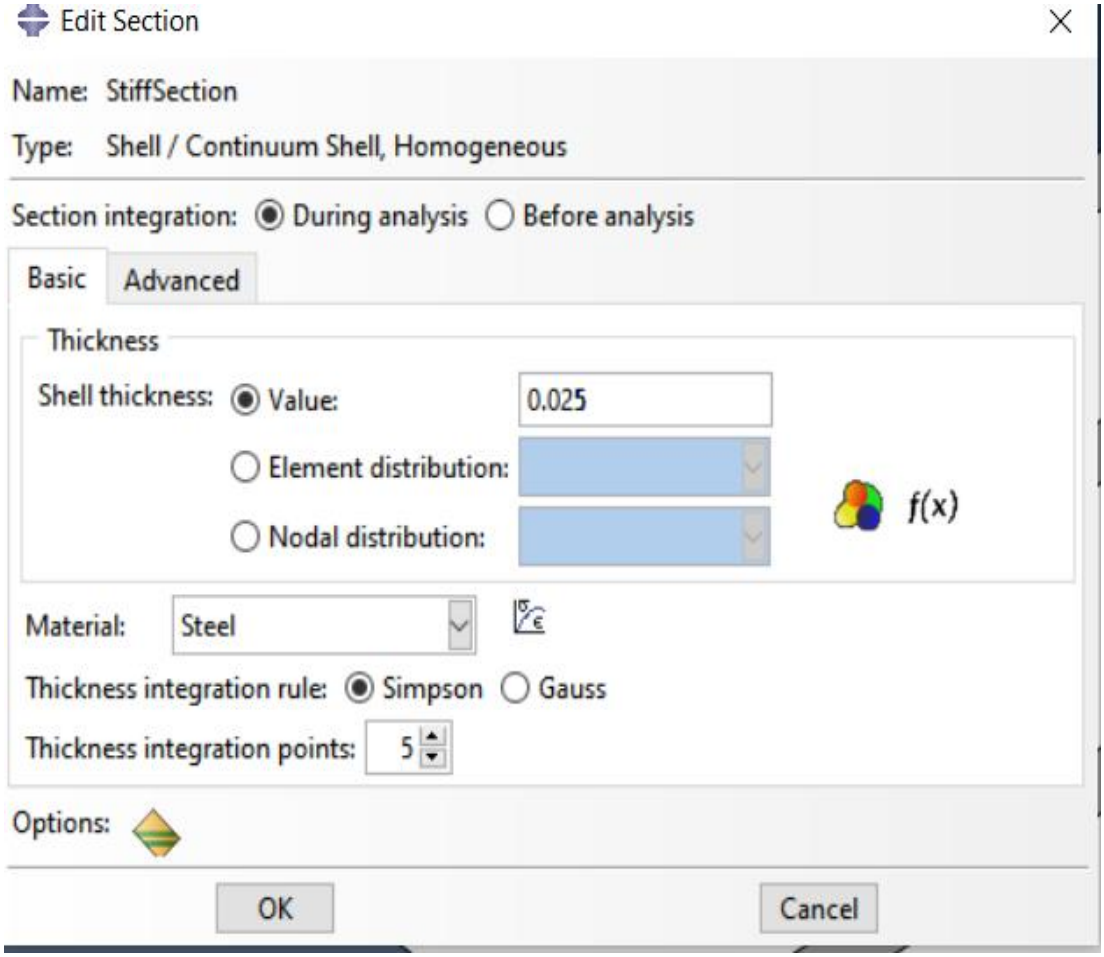

Edit Base Extrusion
End Condition
Type: Blind
Depth: 2
Options
Note: Twist and draft cannot be specified together.
Include twist, pitch: 0 (Dist/Rev)
Include draft, angle: 0 (Degrees)
OK
Cancel

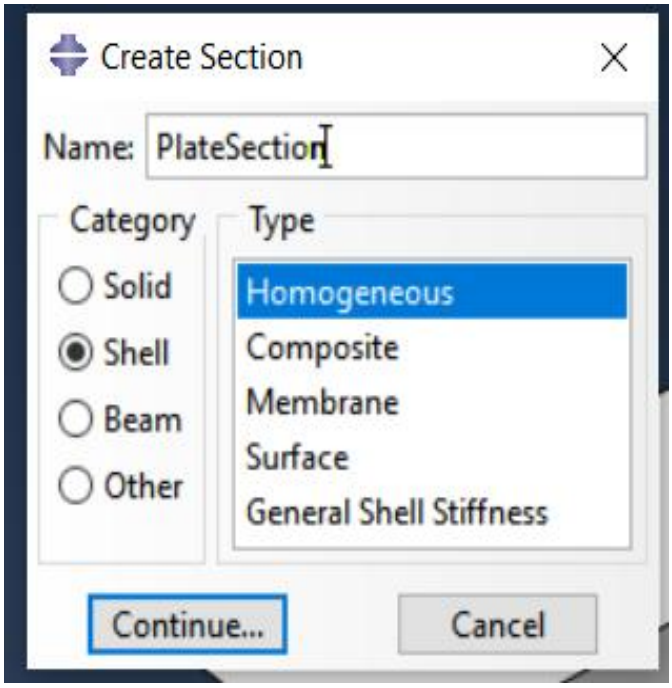

Edit Material
Name: Steel
Description:
Material Behaviors
Density
General Mechanical Thermal Electrical/Magnetic Other
Density
Distribution: Uniform
Use temperature-dependent data
Number of field variables: 0
Data
Mass Density
1 7800
OK
Cancel

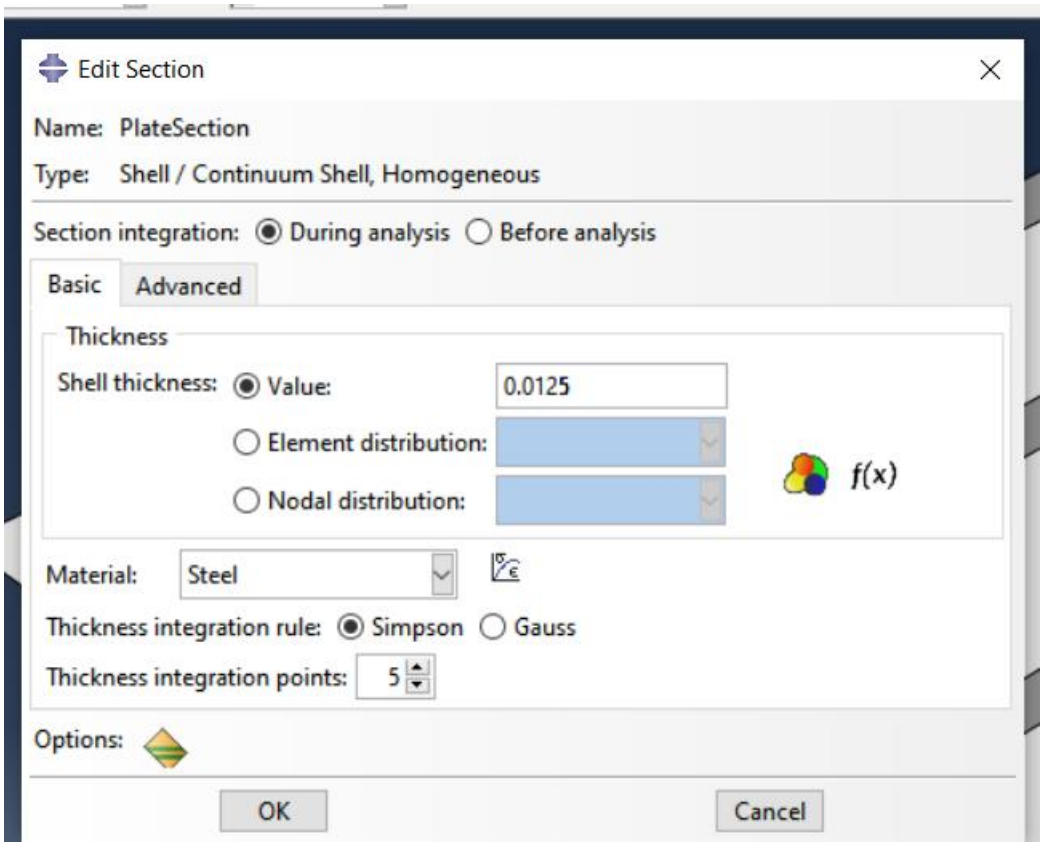

Edit Material
Name: Steel
Description:
Material Behaviors
Density
General
Mechanical
Thermal
Electrical/Magnetic
Other
Elasticity
Plasticity
Damage for Ductile Metals
Damage for Traction Separation Laws
Damage for Fiber-Reinforced Composites
Damage for Elastomers
Deformation Plasticity
Damping
Expansion
Brittle Cracking
Eos
Viscosity
Super Elasticity
Elastic
Hyperelastic
Hyperfoam
Low Density Foam
Hypoelastic
Porous Elastic
Viscoelastic
Density
Distributic
Use ter
Number o
Data
1
OK
Cancel

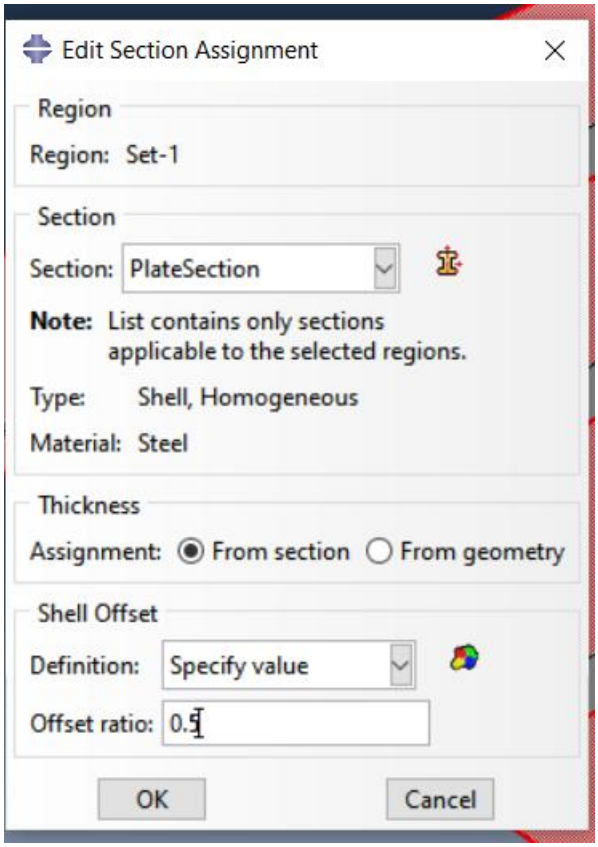

Edit Material
Name: Steel
Description:
Material Behaviors
Density
Elastic
General
Mechanical
Thermal
Electrical/Magnetic
Other
Elastic
Type: Isotropic
Suboptions
Use temperature-dependent data
Number of field variables: 0
Moduli time scale (for viscoelasticity): Long-term
No compression
No tension
Data
Young's Modulus
Poisson's Ratio
1
210e9
0.3
OK
Cancel

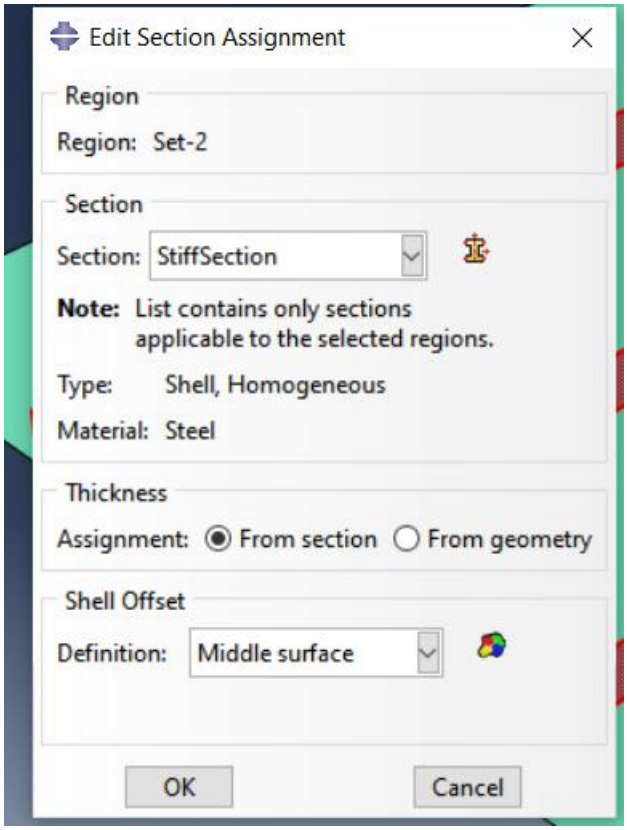

Create Section
Name: StiffSection
Category
Solid
Shell
Beam
Other
Type
Homogeneous
Composite
Membrane
Surface
General Shell Stiffness
Continue...
Cancel

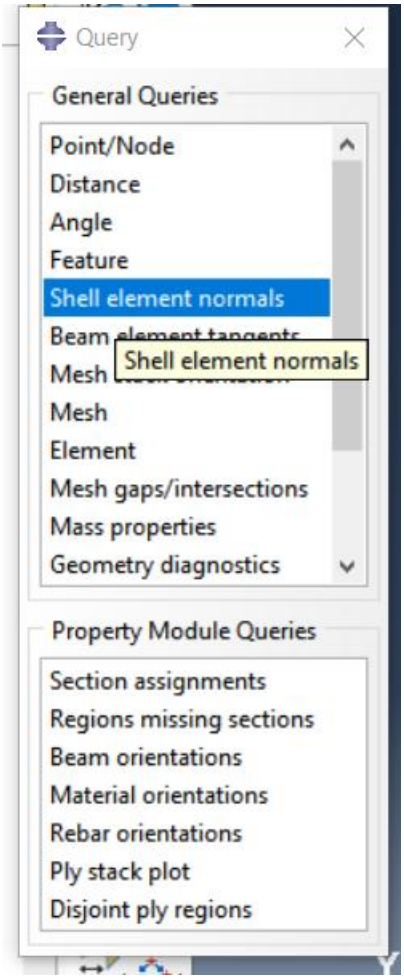

Edit Section
Name: StiffSection
Type: Shell / Continuum Shell, Homogeneous
Section integration: During analysis Before analysis
Basic
Advanced
Thickness
Shell thickness: Value: 0.025
Element distribution:
Nodal distribution:
f(x)
Material: Steel
Thickness integration rule: Simpson Gauss
Thickness integration points: 5
Options:
OK
Cancel

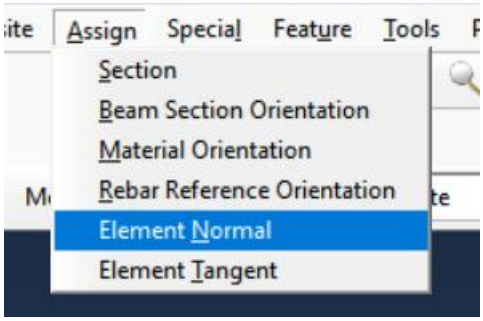

Create Section
Name: PlateSection
Category
Solid
Shell
Beam
Other
Type
Homogeneous
Composite
Membrane
Surface
General Shell Stiffness
Continue...
Cancel

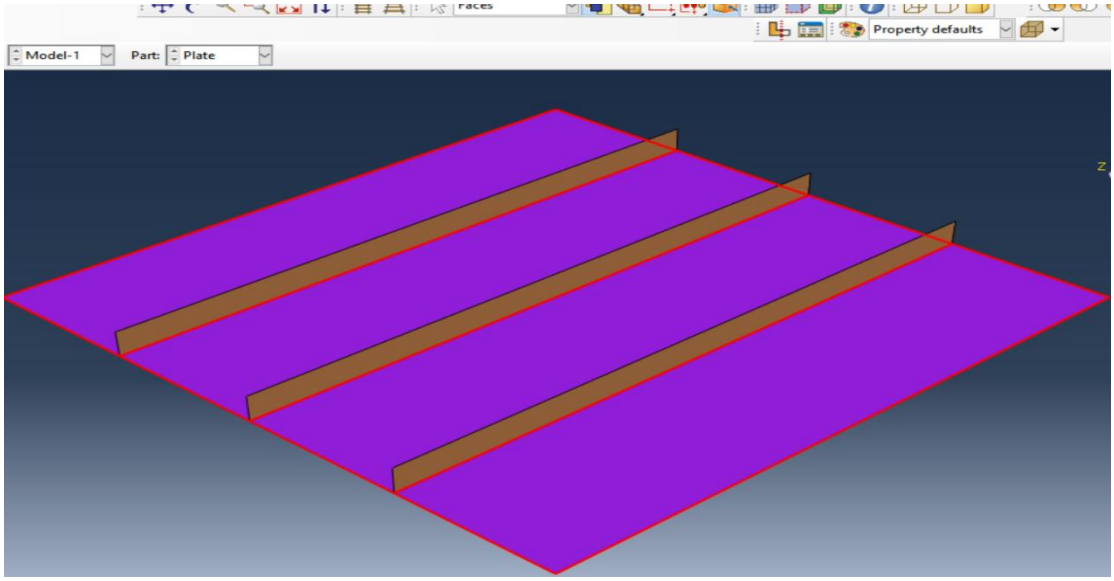

Edit Section
Name: PlateSection
Type: Shell / Continuum Shell, Homogeneous
Section integration: During analysis Before analysis
Basic
Advanced
Thickness
Shell thickness: Value: 0.0125
Element distribution:
Nodal distribution:
f(x)
Material: Steel
Thickness integration rule: Simpson Gauss
Thickness integration points: 5
Options:
OK
Cancel

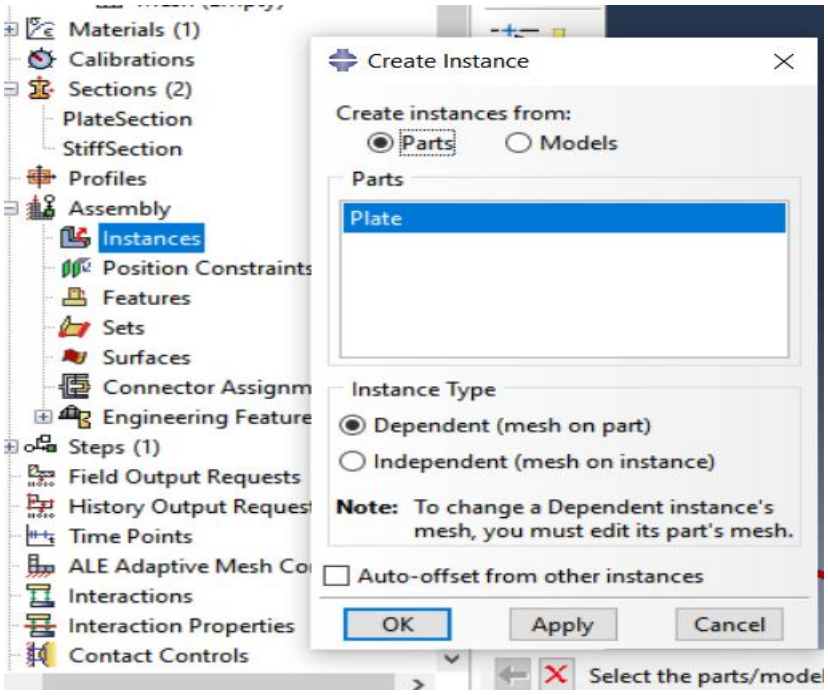

Edit Section Assignment
Region
Region: Set-1
Section
Section: PlateSection
Note: List contains only sections applicable to the selected regions.
Type: Shell, Homogeneous
Material: Steel
Thickness
Assignment: From section From geometry
Shell Offset
Definition: Specify value
Offset ratio: 0.5
OK
Cancel

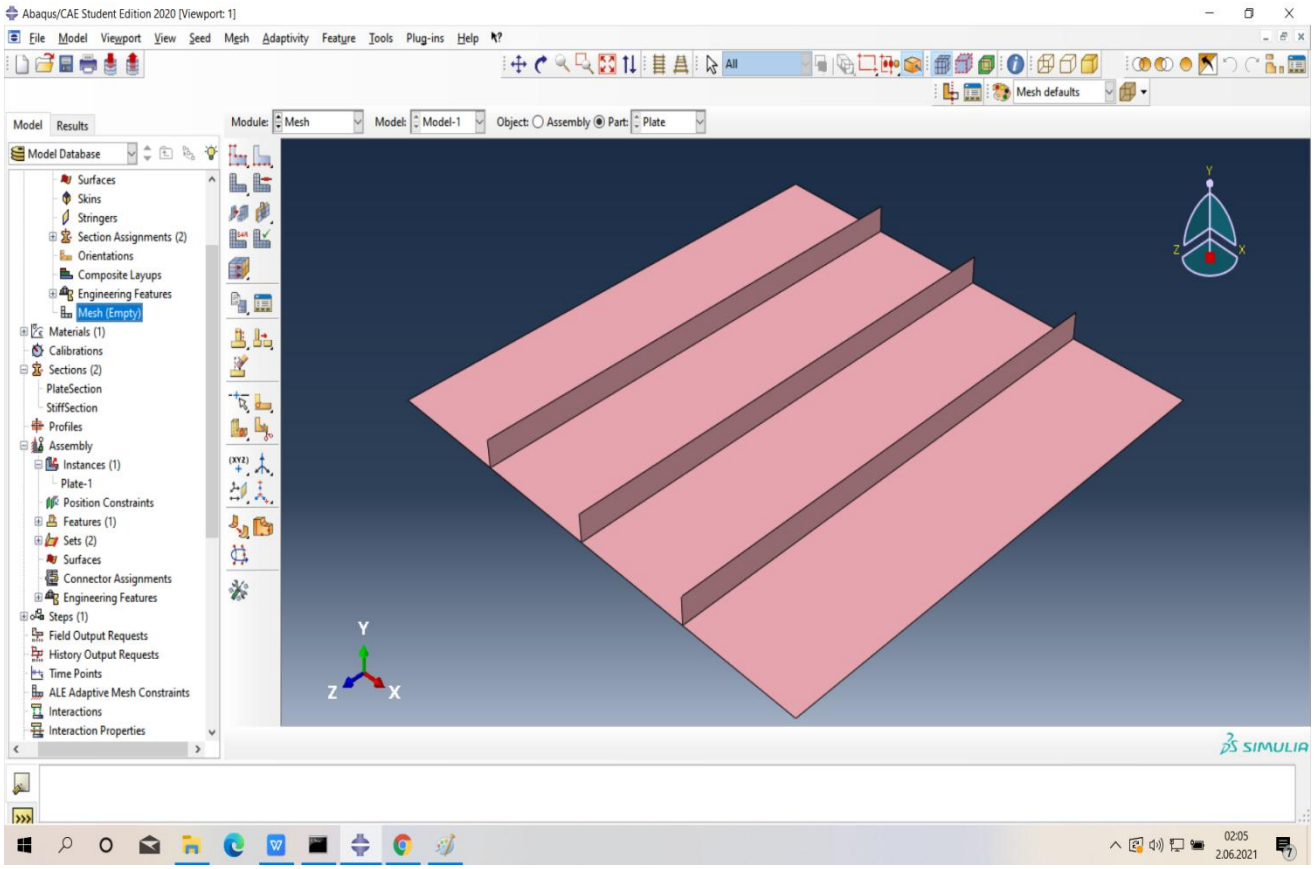

Edit Section Assignment
Region
Region: Set-2
Section
Section: StiffSection
Note: List contains only sections applicable to the selected regions.
Type: Shell, Homogeneous
Material: Steel
Thickness
Assignment: From section From geometry
Shell Offset
Definition: Middle surface
OK
Cancel

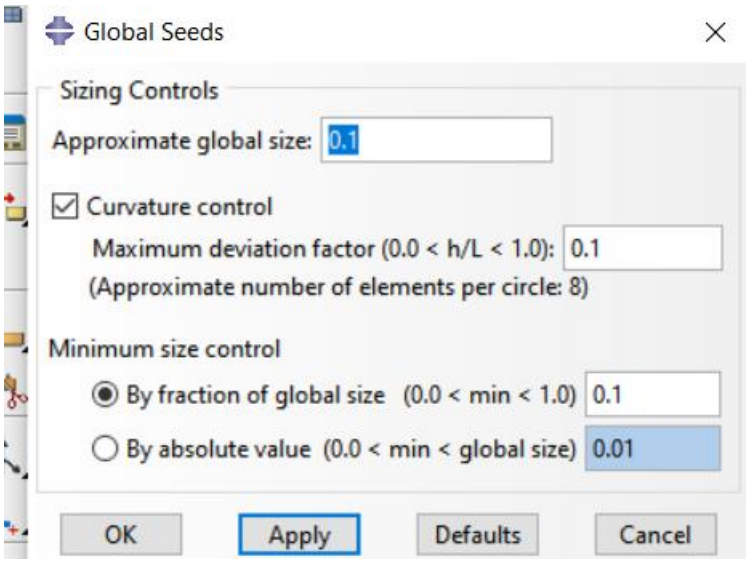

Query
General Queries
Point/Node
Distance
Angle
Feature
Shell element normals
Beam element tangents
Shell element normals
Mesh
Mesh
Element
Mesh gaps/intersections
Mass properties
Geometry diagnostics
Property Module Queries
Section assignments
Regions missing sections
Beam orientations
Material orientations
Rebar orientations
Ply stack plot
Disjoint ply regions

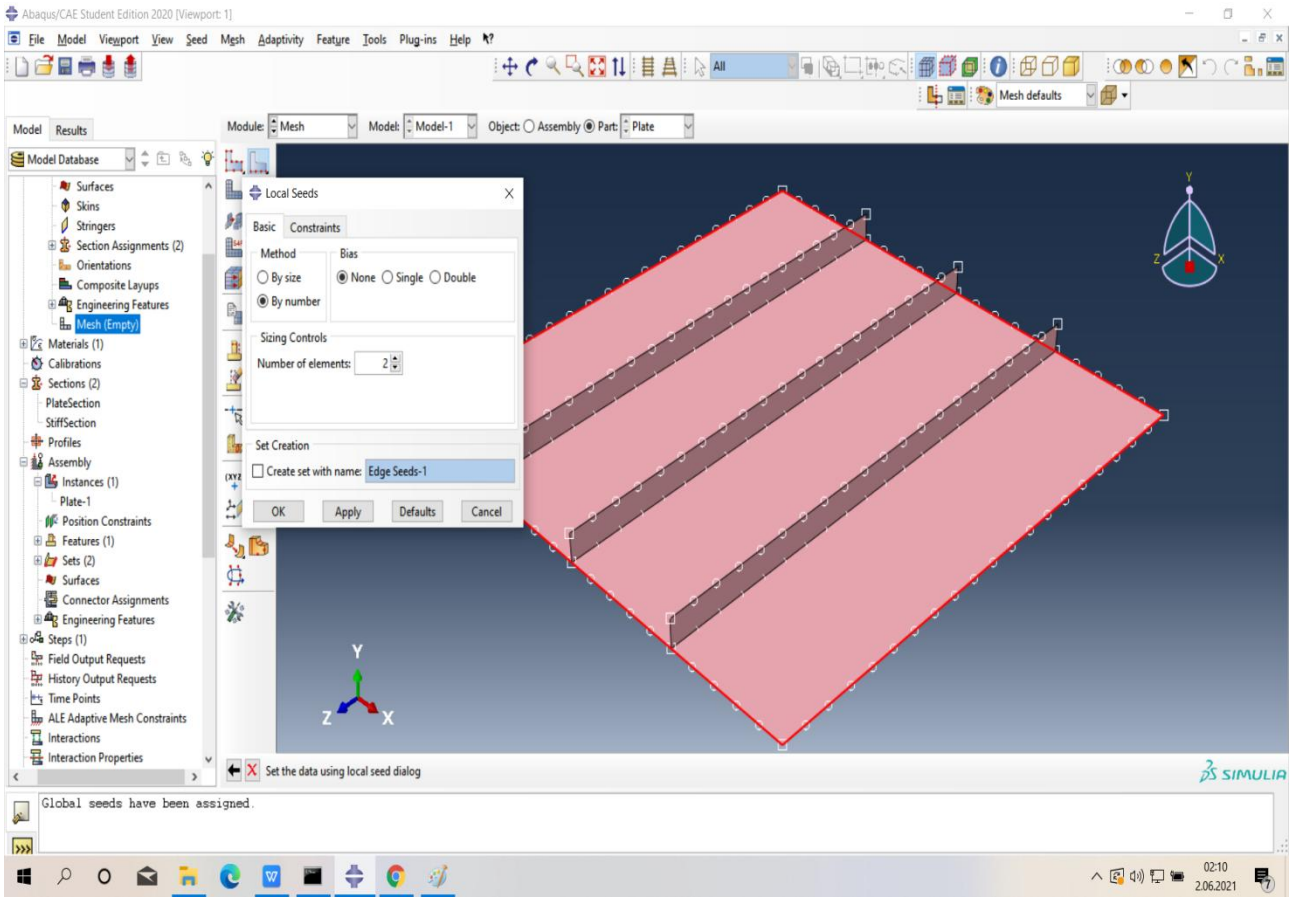

Assign
Special
Feature
Tools
Section
Beam Section Orientation
Material Orientation
Rebar Reference Orientation
Element Normal
Element Tangent

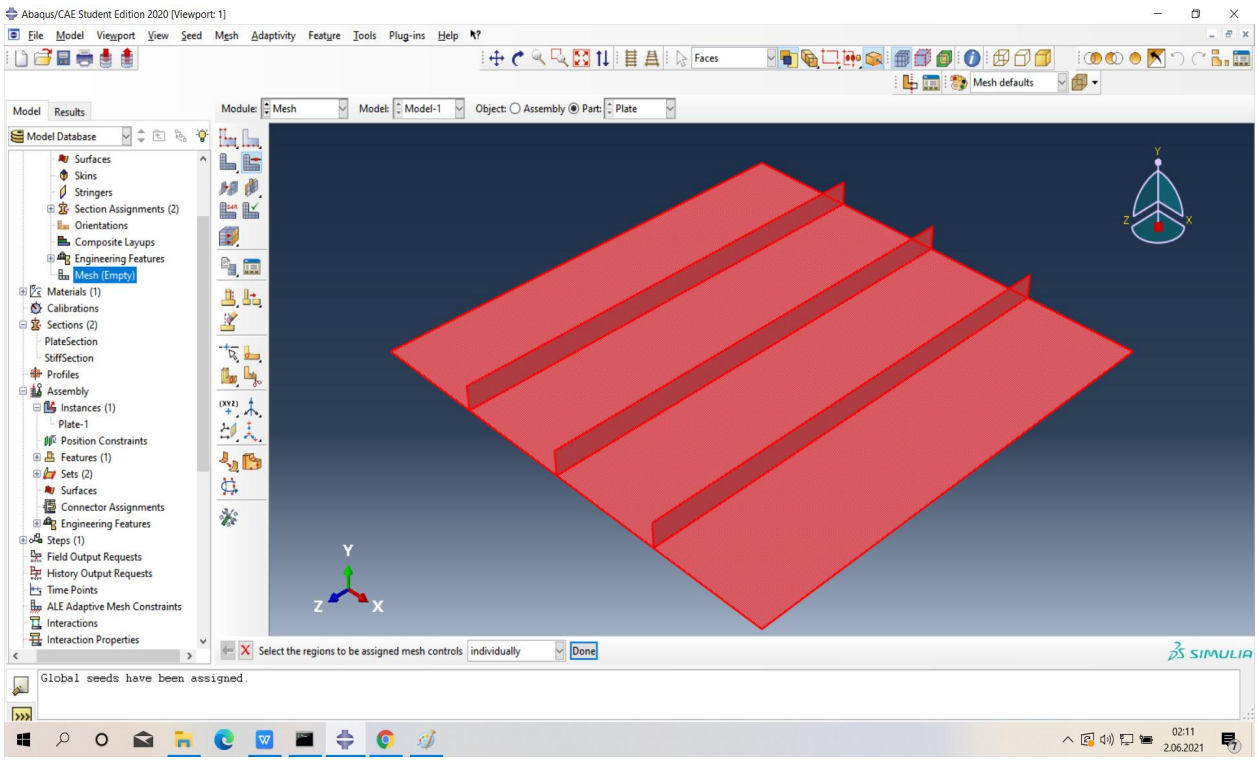

Model-1
Part: Plate
Property defaults

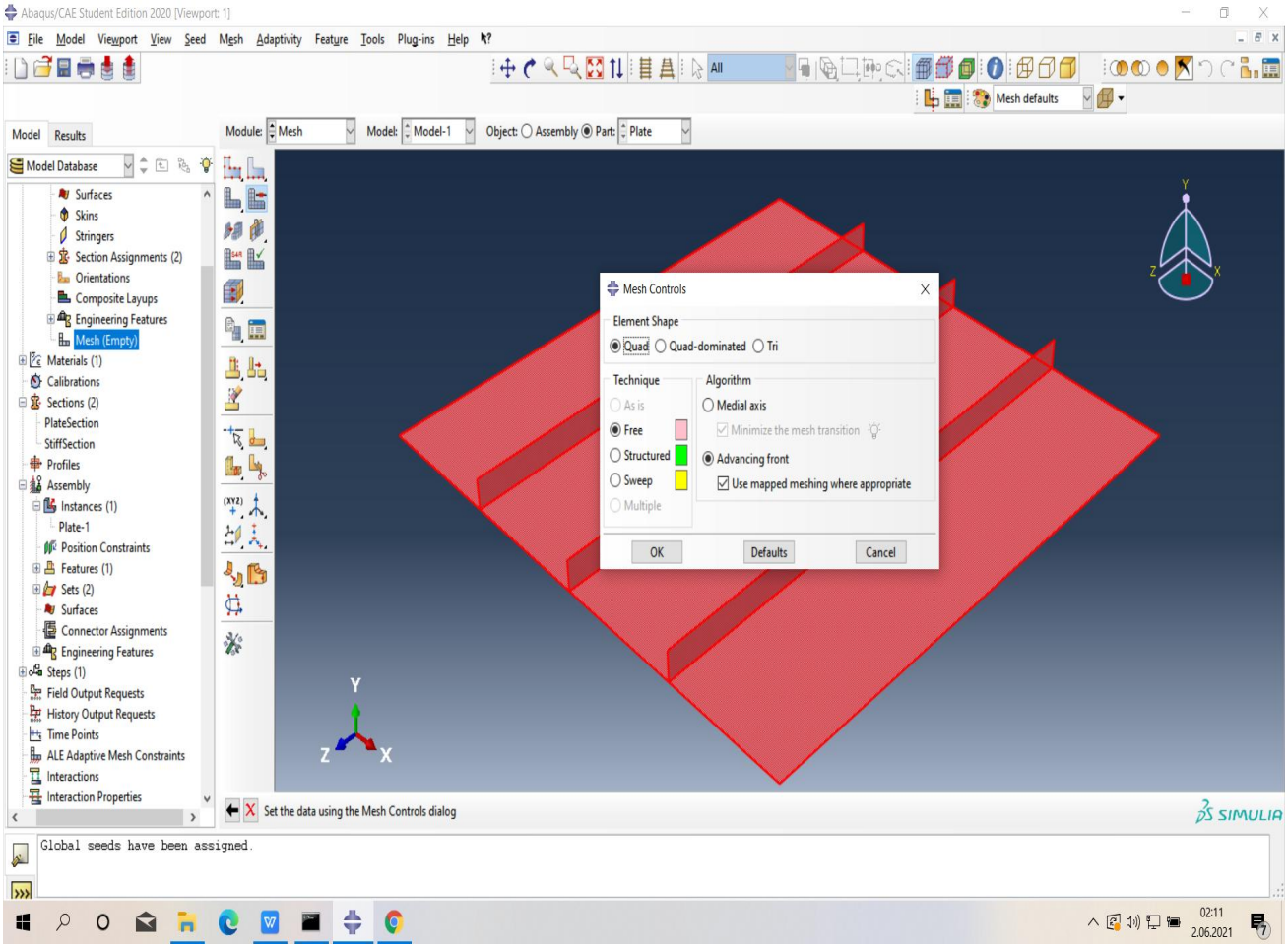

Materials (1)
Calibrations
Sections (2)
PlateSection
StiffSection
Profiles
Assembly
Instances
Position Constraints
Features
Sets
Surfaces
Connector Assignm
Engineering Feature
Steps (1)
Field Output Requests
History Output Reques
Time Points
ALE Adaptive Mesh Co
Interactions
Interaction Properties
Contact Controls
Create Instance
Create instances from:
Parts
Models
Parts
Plate
Instance Type
Dependent (mesh on part)
Independent (mesh on instance)
Note: To change a Dependent instance's mesh, you must edit its part's mesh.
Auto-offset from other instances
OK
Apply
Cancel
Select the parts/model

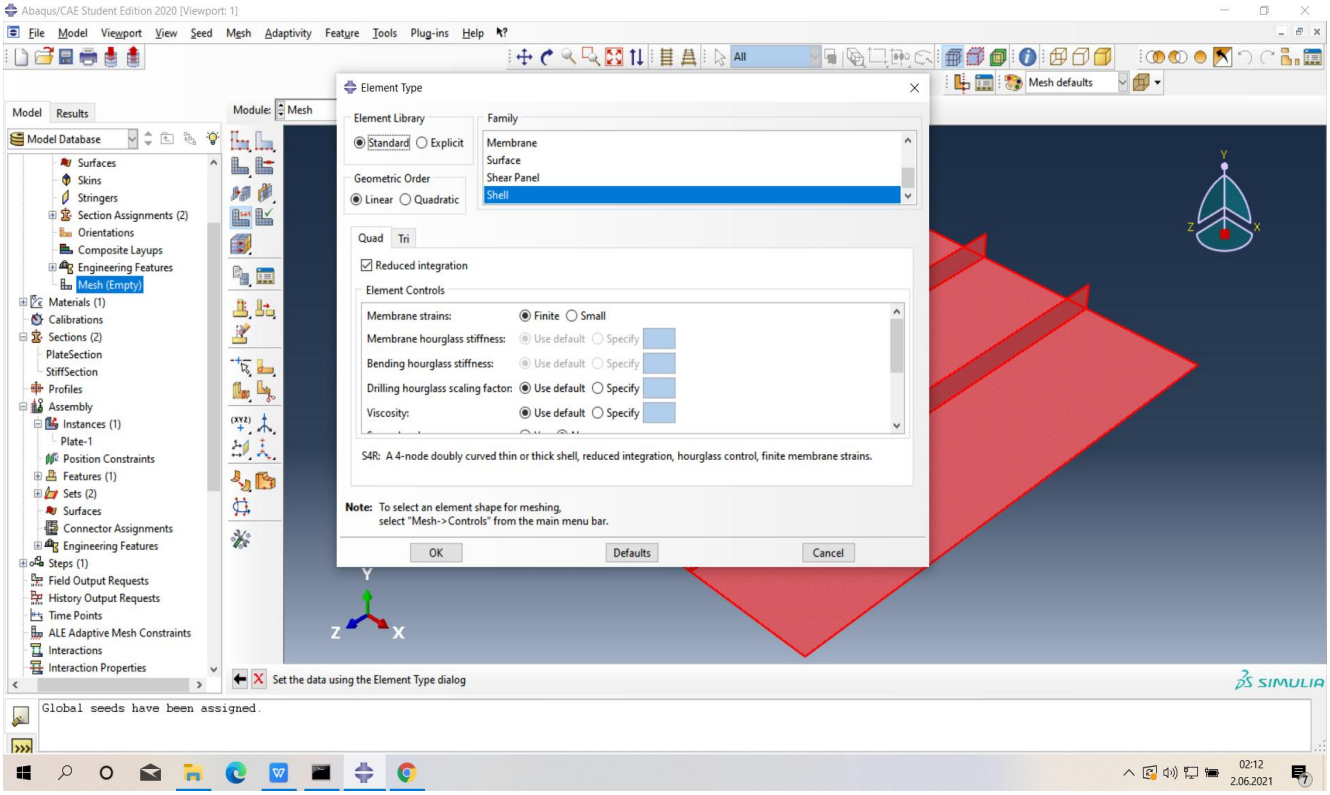

Abaqus/CAE Student Edition 2020 [Viewport: 1]
File Model Viewport View Seed Mesh Adaptivity Feature Tools Plug-ins Help
Mesh defaults
Model Results
Module: Mesh
Model: Model-1
Object: Assembly Part: Plate
Model Database
Surfaces
Skins
Stringers
Section Assignments (2)
Orientations
Composite Layups
Engineering Features
Mesh (Empty)
Materials (1)
Calibrations
Sections (2)
PlateSection
StiffSection
Profiles
Assembly
Instances (1)
Plate-1
Position Constraints
Features (1)
Sets (2)
Surfaces
Connector Assignments
Engineering Features
Steps (1)
Field Output Requests
History Output Requests
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Y
Z
X
SIMULIA
02:05
2.06.2021

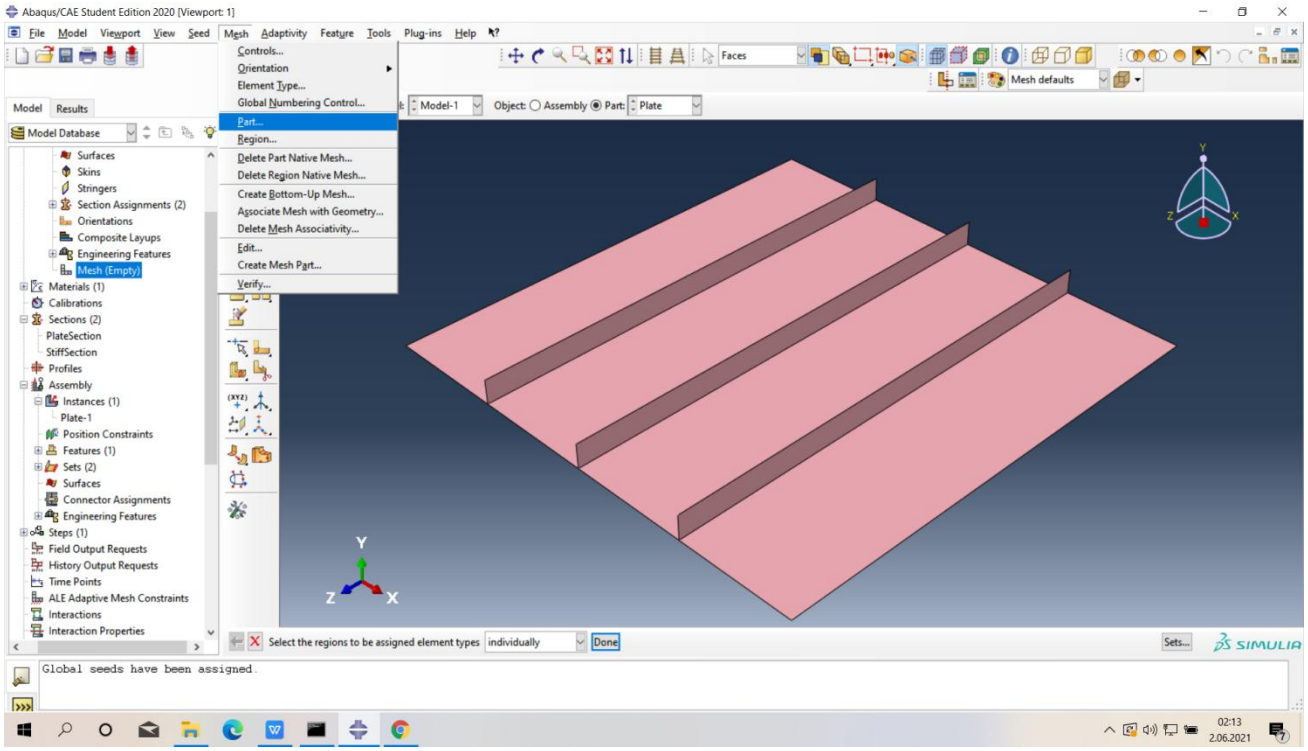

Global Seeds
Sizing Controls
Approximate global size: 0.1
Curvature control
Maximum deviation factor (0.0 < h/L < 1.0): 0.1
(Approximate number of elements per circle: 8)
Minimum size control
By fraction of global size (0.0 < min < 1.0) 0.1
By absolute value (0.0 < min < global size) 0.01
OK
Apply
Defaults
Cancel

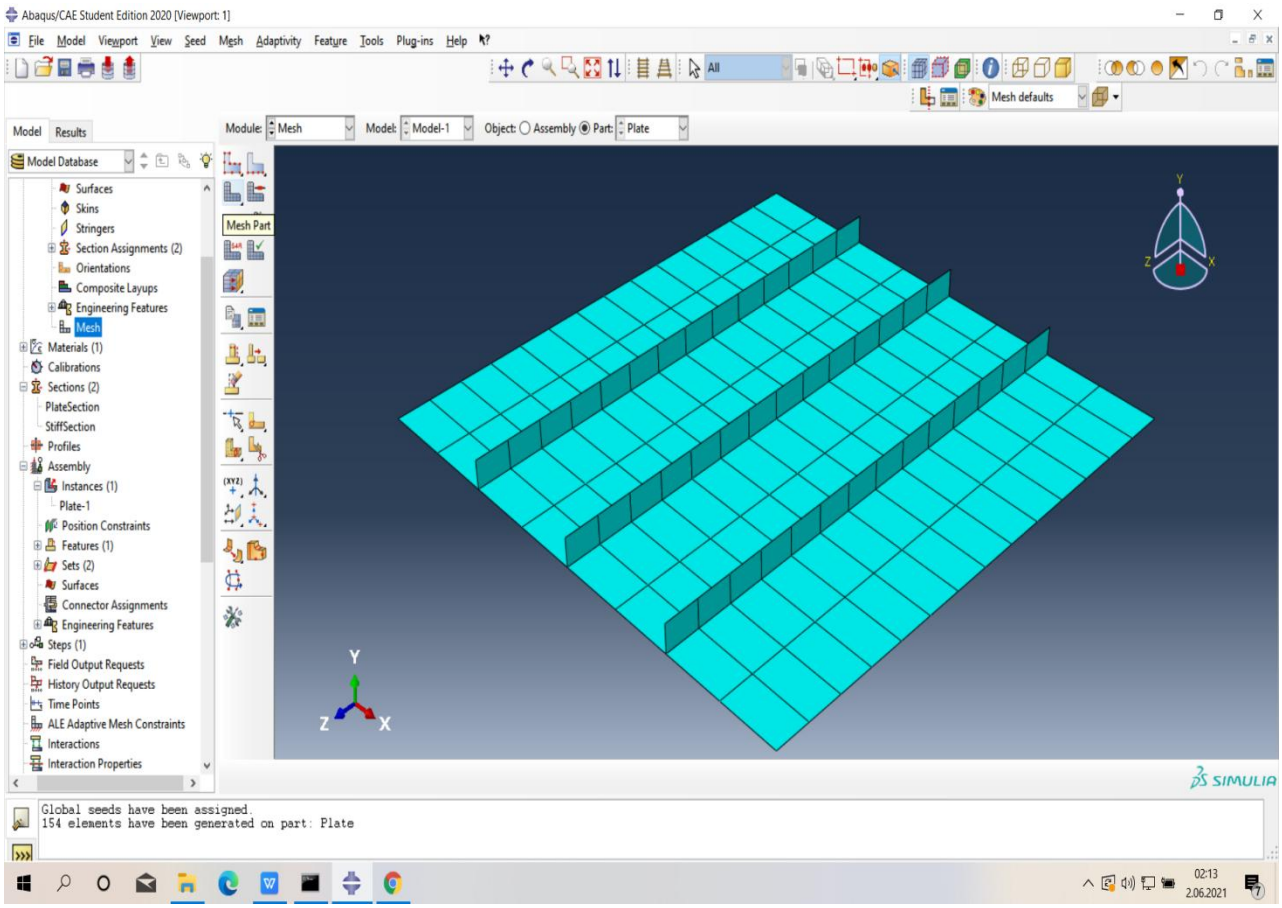

Abaqus/CAE Student Edition 2020 [Viewport: 1]
File Model Viewport View Seed Mesh Adaptivity Feature Tools Plug-ins Help
Mesh defaults
Model Results
Module: Mesh Model: Model-1 Object: Assembly Part: Plate
Model Database
Surfaces
Skins
Stringers
Section Assignments (2)
Orientations
Composite Layups
Engineering Features
Mesh (Empty)
Materials (1)
Calibrations
Sections (2)
PlateSection
StiffSection
Profiles
Assembly
Instances (1)
Plate-1
Position Constraints
Features (1)
Sets (2)
Surfaces
Connector Assignments
Engineering Features
Steps (1)
Field Output Requests
History Output Requests
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Local Seeds
Basic Constraints
Method
By size
By number
Bias
None Single Double
Sizing Controls
Number of elements: 2
Set Creation
Create set with name: Edge Seeds-1
OK Apply Defaults Cancel
Set the data using local seed dialog
SIMULIA
Global seeds have been assigned.
02:10
2.06.2021

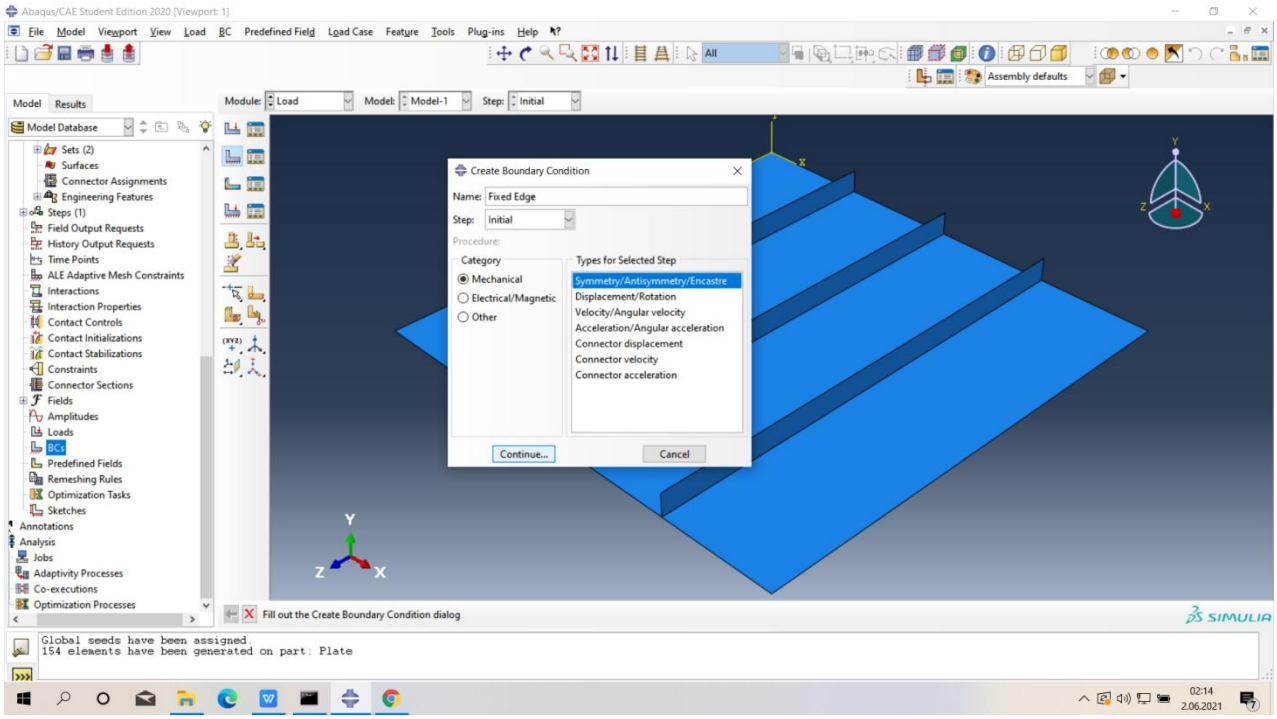

Abaqus/CAE Student Edition 2020 [Viewport: 1]
File Model Viewport View Seed Mesh Adaptivity Feature Tools Plug-ins Help
Faces
Mesh defaults
Model Results
Module: Mesh Model: Model-1 Object: Assembly Part: Plate
Model Database
Surfaces
Skins
Stringers
Section Assignments (2)
Orientations
Composite Layups
Engineering Features
Mesh (Empty)
Materials (1)
Calibrations
Sections (2)
PlateSection
StiffSection
Profiles
Assembly
Instances (1)
Plate-1
Position Constraints
Features (1)
Sets (2)
Surfaces
Connector Assignments
Engineering Features
Steps (1)
Field Output Requests
History Output Requests
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Select the regions to be assigned mesh controls
individually
Done
SIMULIA
Global seeds have been assigned.
02:11
2.06.2021

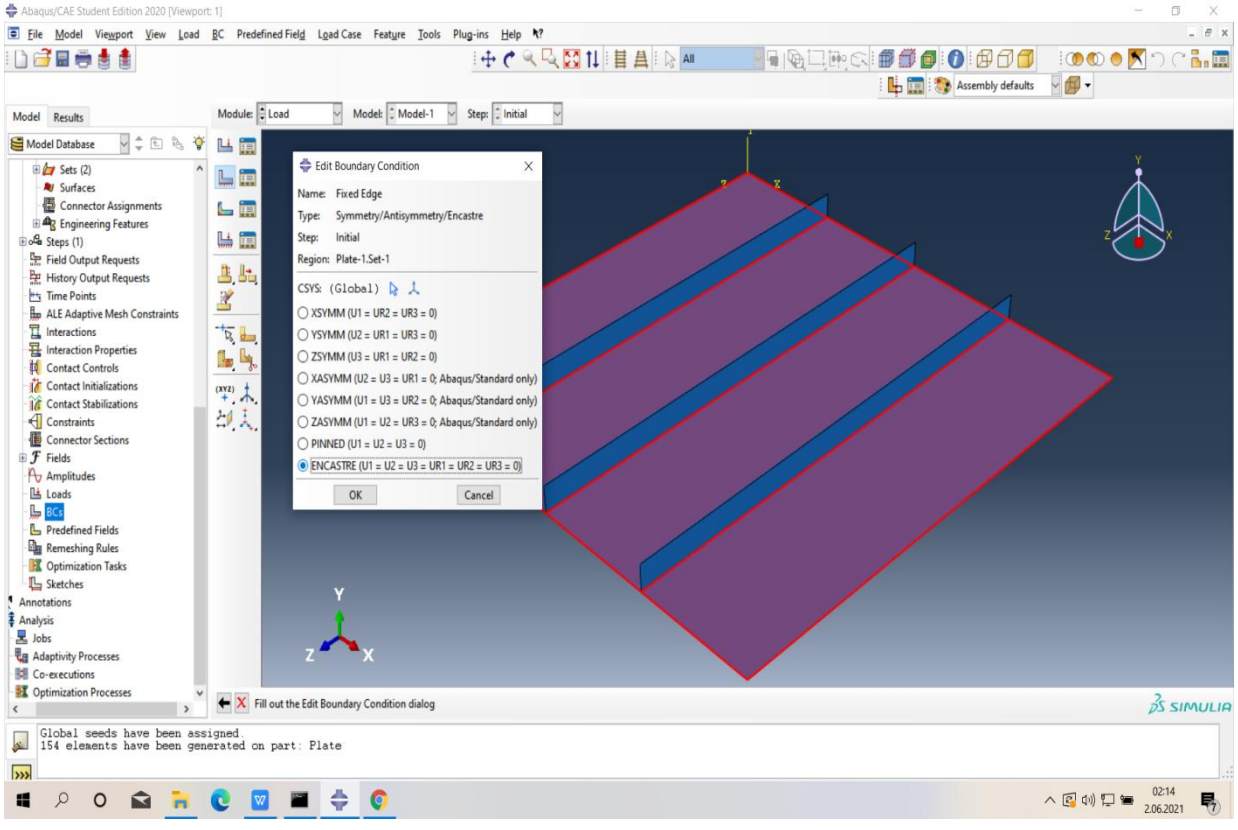

Abaqus/CAE Student Edition 2020 [Viewport: 1]
File Model Viewport View Seed Mesh Adaptivity Feature Tools Plug-ins Help
Model Results
Module: Mesh Model: Model-1 Object: Assembly Part: Plate
Model Database
Surfaces
Skins
Stringers
Section Assignments (2)
Orientations
Composite Layups
Engineering Features
Mesh (Empty)
Materials (1)
Calibrations
Sections (2)
PlateSection
StiffSection
Profiles
Assembly
Instances (1)
Plate-1
Position Constraints
Features (1)
Sets (2)
Surfaces
Connector Assignments
Engineering Features
Steps (1)
Field Output Requests
History Output Requests
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Mesh Controls
Element Shape
Quad Quad-dominated Tri
Technique
As is
Free
Structured
Sweep
Multiple
Algorithm
Medial axis
Minimize the mesh transition
Advancing front
Use mapped meshing where appropriate
OK Defaults Cancel
Set the data using the Mesh Controls dialog
SIMULIA
Global seeds have been assigned.
02:11
2.06.2021

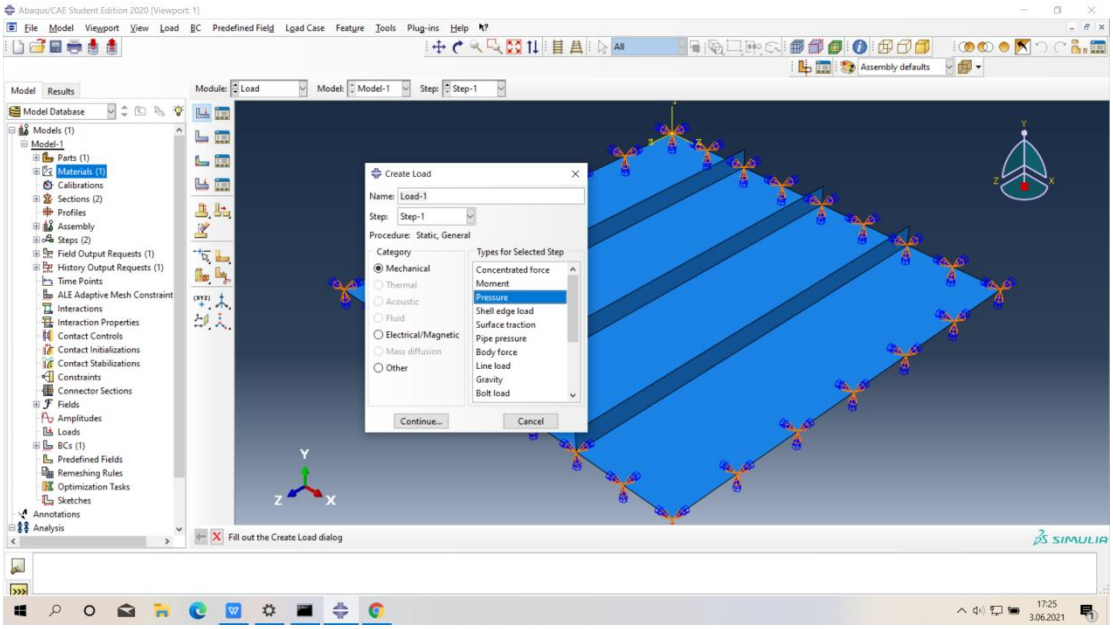

Abaqus/CAE Student Edition 2020 [Viewport: 1]
File Model Viewport View Seed Mesh Adaptivity Feature Tools Plug-ins Help
Model Results
Module: Mesh
Model Database
Surfaces
Skins
Stringers
Section Assignments (2)
Orientations
Composite Layups
Engineering Features
Mesh (Empty)
Materials (1)
Calibrations
Sections (2)
PlateSection
StiffSection
Profiles
Assembly
Instances (1)
Plate-1
Position Constraints
Features (1)
Sets (2)
Surfaces
Connector Assignments
Engineering Features
Steps (1)
Field Output Requests
History Output Requests
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Element Type
Element Library
Standard Explicit
Geometric Order
Linear Quadratic
Family
Membrane
Surface
Shear Panel
Shell
Quad Tri
Reduced integration
Element Controls
Membrane strains: Finite Small
Membrane hourglass stiffness: Use default Specify
Bending hourglass stiffness: Use default Specify
Drilling hourglass scaling factor: Use default Specify
Viscosity: Use default Specify
S4R: A 4-node doubly curved thin or thick shell, reduced integration, hourglass control, finite membrane strains.
Note: To select an element shape for meshing, select "Mesh->Controls" from the main menu bar.
OK Defaults Cancel
Set the data using the Element Type dialog
SIMULIA
Global seeds have been assigned.
02:12
2.06.2021

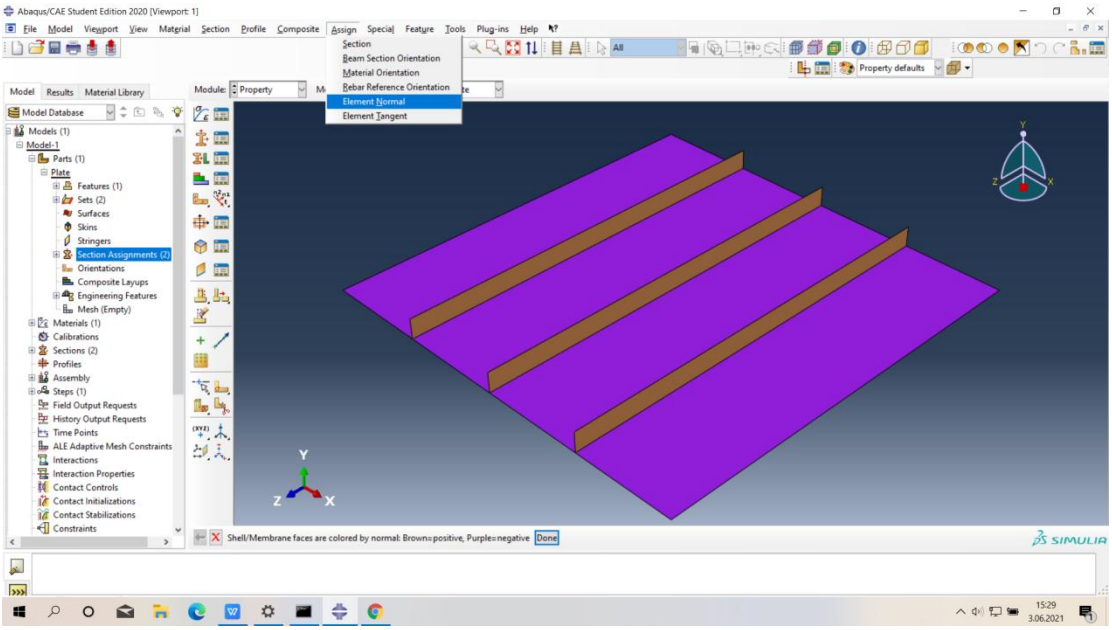

Abaqus/CAE Student Edition 2020 [Viewport: 1]
File Model Viewport View Seed Mesh Adaptivity Feature Tools Plug-ins Help
Controls...
Orientation
Element Type...
Global Numbering Control...
Part...
Region...
Delete Part Native Mesh...
Delete Region Native Mesh...
Create Bottom-Up Mesh...
Associate Mesh with Geometry...
Delete Mesh Associativity...
Edit...
Create Mesh Part...
Verify...
Model Results
Model Database
Surfaces
Skins
Stringers
Section Assignments (2)
Orientations
Composite Layups
Engineering Features
Mesh (Empty)
Materials (1)
Calibrations
Sections (2)
PlateSection
StiffSection
Profiles
Assembly
Instances (1)
Plate-1
Position Constraints
Features (1)
Sets (2)
Surfaces
Connector Assignments
Engineering Features
Steps (1)
Field Output Requests
History Output Requests
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Faces
Mesh defaults
Model-1
Object: Assembly Part: Plate
Select the regions to be assigned element types
individually
Done
Sets...
SIMULIA
Global seeds have been assigned.
02:13
2.06.2021

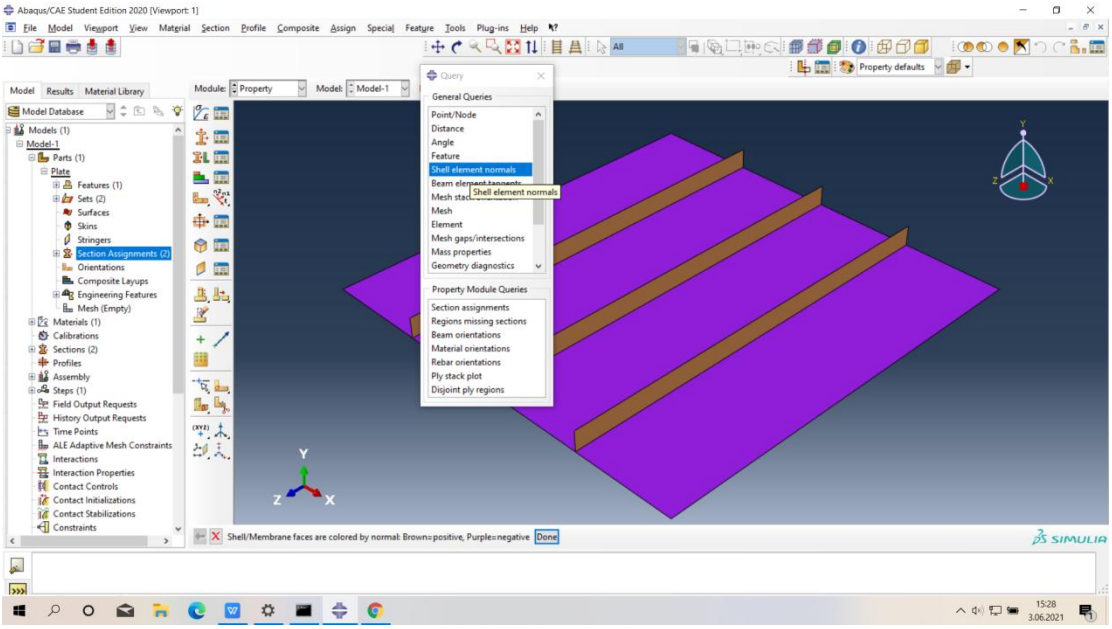

Abaqus/CAE Student Edition 2020 [Viewport: 1]
File Model Viewport View Seed Mesh Adaptivity Feature Tools Plug-ins Help
Model Results
Model Database
Surfaces
Skins
Stringers
Section Assignments (2)
Orientations
Composite Layups
Engineering Features
Mesh
Materials (1)
Calibrations
Sections (2)
PlateSection
StiffSection
Profiles
Assembly
Instances (1)
Plate-1
Position Constraints
Features (1)
Sets (2)
Surfaces
Connector Assignments
Engineering Features
Steps (1)
Field Output Requests
History Output Requests
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Module: Mesh
Model: Model-1
Object: Assembly Part: Plate
Mesh Part
Mesh defaults
SIMULIA
Global seeds have been assigned.
154 elements have been generated on part: Plate
02:13
2.06.2021

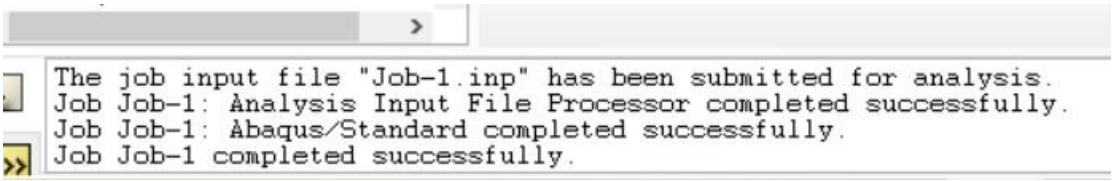

Abaqus/CAE Student Edition 2020 [Viewport: 1]
File Model Viewport View Load BC Predefined Field Load Case Feature Tools Plug-ins Help
Assembly defaults
Model Results
Module: Load Model: Model-1 Step: Initial
Model Database
Sets (2)
Surfaces
Connector Assignments
Engineering Features
Steps (1)
Field Output Requests
History Output Requests
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Contact Controls
Contact Initializations
Contact Stabilizations
Constraints
Connector Sections
Fields
Amplitudes
Loads
BCs
Predefined Fields
Remeshing Rules
Optimization Tasks
Sketches
Annotations
Analysis
Jobs
Adaptivity Processes
Co-executions
Optimization Processes
Create Boundary Condition
Name: Fixed Edge
Step: Initial
Procedure:
Category
Mechanical
Electrical/Magnetic
Other
Types for Selected Step
Symmetry/Antisymmetry/Encastre
Displacement/Rotation
Velocity/Angular velocity
Acceleration/Angular acceleration
Connector displacement
Connector velocity
Connector acceleration
Continue...
Cancel
Fill out the Create Boundary Condition dialog
SIMULIA
Global seeds have been assigned.
154 elements have been generated on part: Plate
02:14
2.06.2021


Abaqus/CAE Student Edition 2020 [Viewport: 1]
File Model Viewport View Load BC Predefined Field Load Case Feature Tools Plug-ins Help
Assembly defaults
Model Results
Module: Load Model: Model-1 Step: Initial
Model Database
Sets (2)
Surfaces
Connector Assignments
Engineering Features
Steps (1)
Field Output Requests
History Output Requests
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Contact Controls
Contact Initializations
Contact Stabilizations
Constraints
Connector Sections
Fields
Amplitudes
Loads
BCs
Predefined Fields
Remeshing Rules
Optimization Tasks
Sketches
Annotations
Analysis
Jobs
Adaptivity Processes
Co-executions
Optimization Processes
Edit Boundary Condition
Name: Fixed Edge
Type: Symmetry/Antisymmetry/Encastre
Step: Initial
Region: Plate-1.Set-1
CSYS: (Global)
XSYMM (U1 = UR2 = UR3 = 0)
YSYMM (U2 = UR1 = UR3 = 0)
ZSYMM (U3 = UR1 = UR2 = 0)
XASYMM (U2 = U3 = UR1 = 0; Abaqus/Standard only)
YASYMM (U1 = U3 = UR2 = 0; Abaqus/Standard only)
ZASYMM (U1 = U2 = UR3 = 0; Abaqus/Standard only)
PINNED (U1 = U2 = U3 = 0)
ENCASTRE (U1 = U2 = U3 = UR1 = UR2 = UR3 = 0)
OK
Cancel
Fill out the Edit Boundary Condition dialog
SIMULIA
Global seeds have been assigned.
154 elements have been generated on part: Plate
02:14
2.06.2021


Abaqus/CAE Student Edition 2020 [Viewport: 1]
File Model Viewport View Load BC Predefined Field Load Case Feature Tools Plug-ins Help
Assembly defaults
Model Results
Module: Load Model: Model-1 Step: Step-1
Model Database
Models (1)
Model-1
Parts (1)
Materials (1)
Calibrations
Sections (2)
Profiles
Assembly
Steps (2)
Field Output Requests (1)
History Output Requests (1)
Time Points
ALE Adaptive Mesh Constraint
Interactions
Interaction Properties
Contact Controls
Contact Initializations
Contact Stabilizations
Constraints
Connector Sections
Fields
Amplitudes
Loads
BCs (1)
Predefined Fields
Remeshing Rules
Optimization Tasks
Sketches
Annotations
Analysis
Create Load
Name: Load-1
Step: Step-1
Procedure: Static, General
Category
Mechanical
Thermal
Acoustic
Fluid
Electrical/Magnetic
Mass diffusion
Other
Types for Selected Step
Concentrated force
Moment
Pressure
Shell edge load
Surface traction
Pipe pressure
Body force
Line load
Gravity
Bolt load
Continue...
Cancel
Fill out the Create Load dialog
SIMULIA
17:25
3.06.2021


Abaqus/CAE Student Edition 2020 [Viewport: 1]
File Model Viewport View Material Section Profile Composite Assign Special Feature Tools Plug-ins Help
Section
Beam Section Orientation
Material Orientation
Rebar Reference Orientation
Element Normal
Element Tangent
Property defaults
Model Results Material Library
Module: Property
Model Database
Models (1)
Model-1
Parts (1)
Plate
Features (1)
Sets (2)
Surfaces
Skins
Stringers
Section Assignments (2)
Orientations
Composite Layups
Engineering Features
Mesh (Empty)
Materials (1)
Calibrations
Sections (2)
Profiles
Assembly
Steps (1)
Field Output Requests
History Output Requests
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Contact Controls
Contact Initializations
Contact Stabilizations
Constraints
Shell/Membrane faces are colored by normal: Brown=positive, Purple=negative
Done
SIMULIA
15:29
3.06.2021


Abaqus/CAE Student Edition 2020 [Viewport: 1]
File Model Viewport View Material Section Profile Composite Assign Special Feature Tools Plug-ins Help
Model Results Material Library
Model Database
Models (1)
Model-1
Parts (1)
Plate
Features (1)
Sets (2)
Surfaces
Skins
Stringers
Section Assignments (2)
Orientations
Composite Layups
Engineering Features
Mesh (Empty)
Materials (1)
Calibrations
Sections (2)
Profiles
Assembly
Steps (1)
Field Output Requests
History Output Requests
Time Points
ALE Adaptive Mesh Constraints
Interactions
Interaction Properties
Contact Controls
Contact Initializations
Contact Stabilizations
Constraints
Module: Property
Model: Model-1
Query
General Queries
Point/Node
Distance
Angle
Feature
Shell element normals
Mesh
Element
Mesh gaps/intersections
Mass properties
Geometry diagnostics
Property Module Queries
Section assignments
Regions missing sections
Beam orientations
Material orientations
Rebar orientations
Ply stack plot
Disjoint ply regions
Shell/Membrane faces are colored by normal: Brown=positive, Purple=negative
Done
SIMULIA
15:28
3.06.2021


The job input file "Job-1.inp" has been submitted for analysis.
Job Job-1: Analysis Input File Processor completed successfully.
Job Job-1: Abaqus/Standard completed successfully.
Job Job-1 completed successfully.