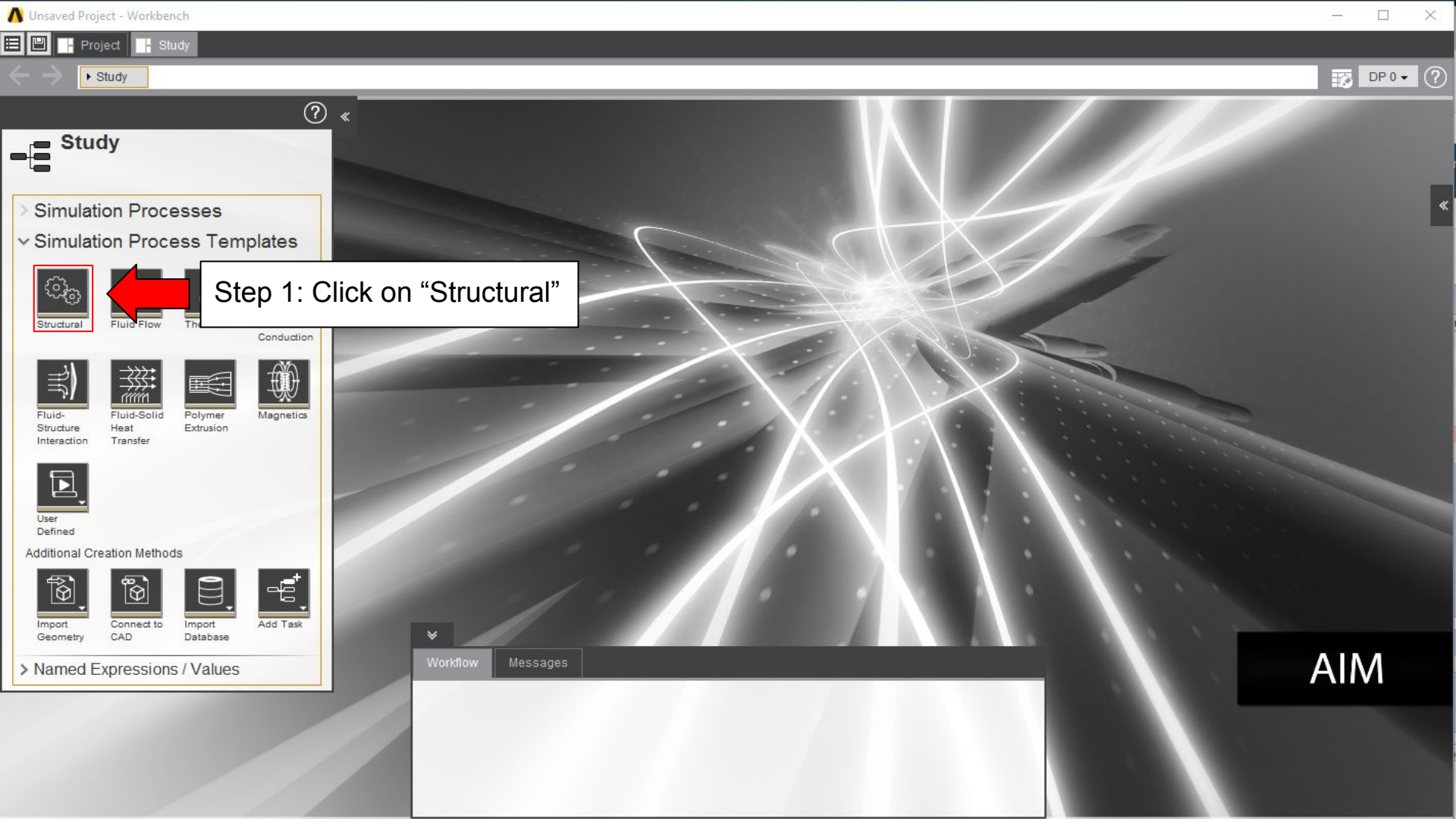
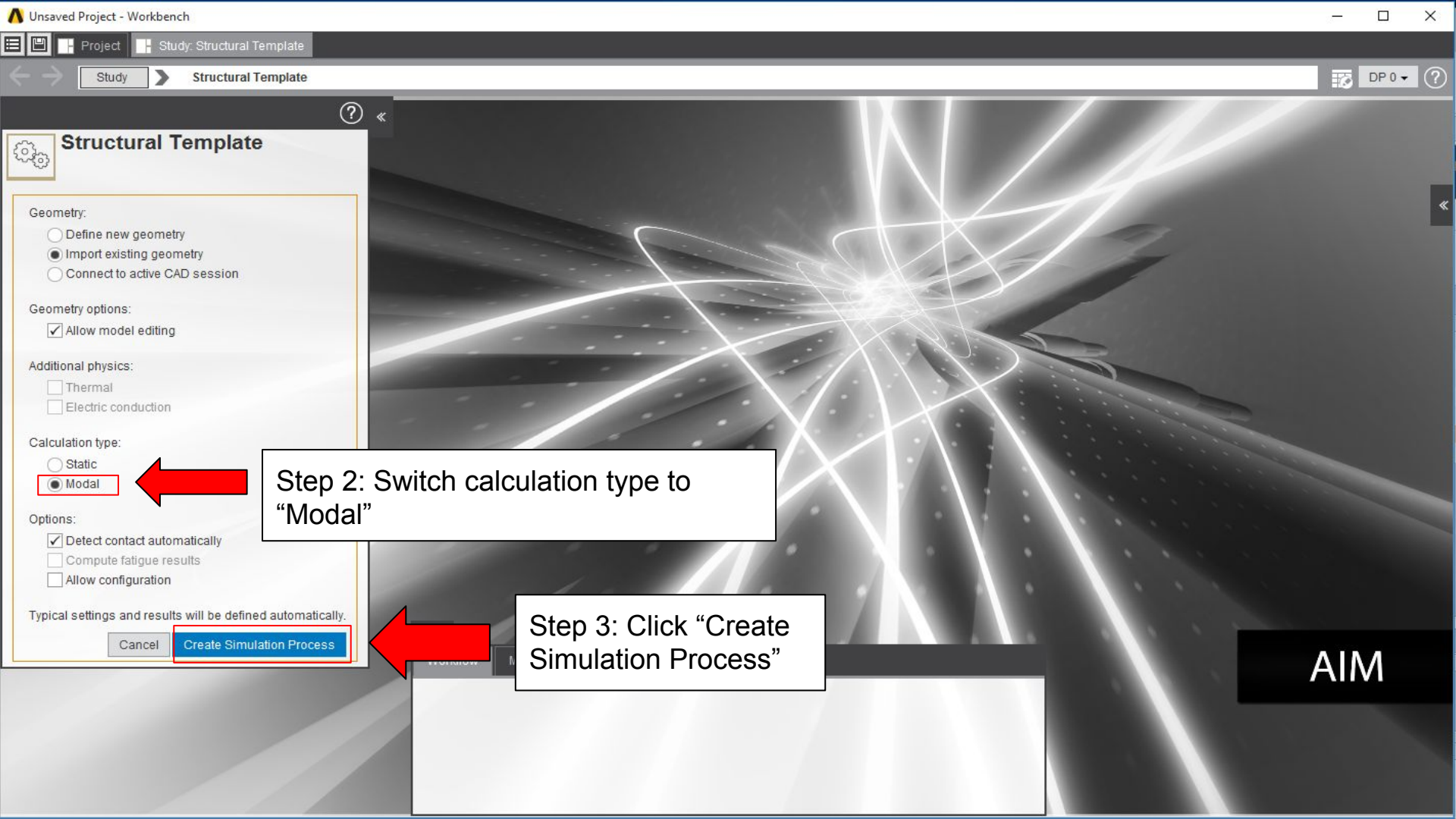
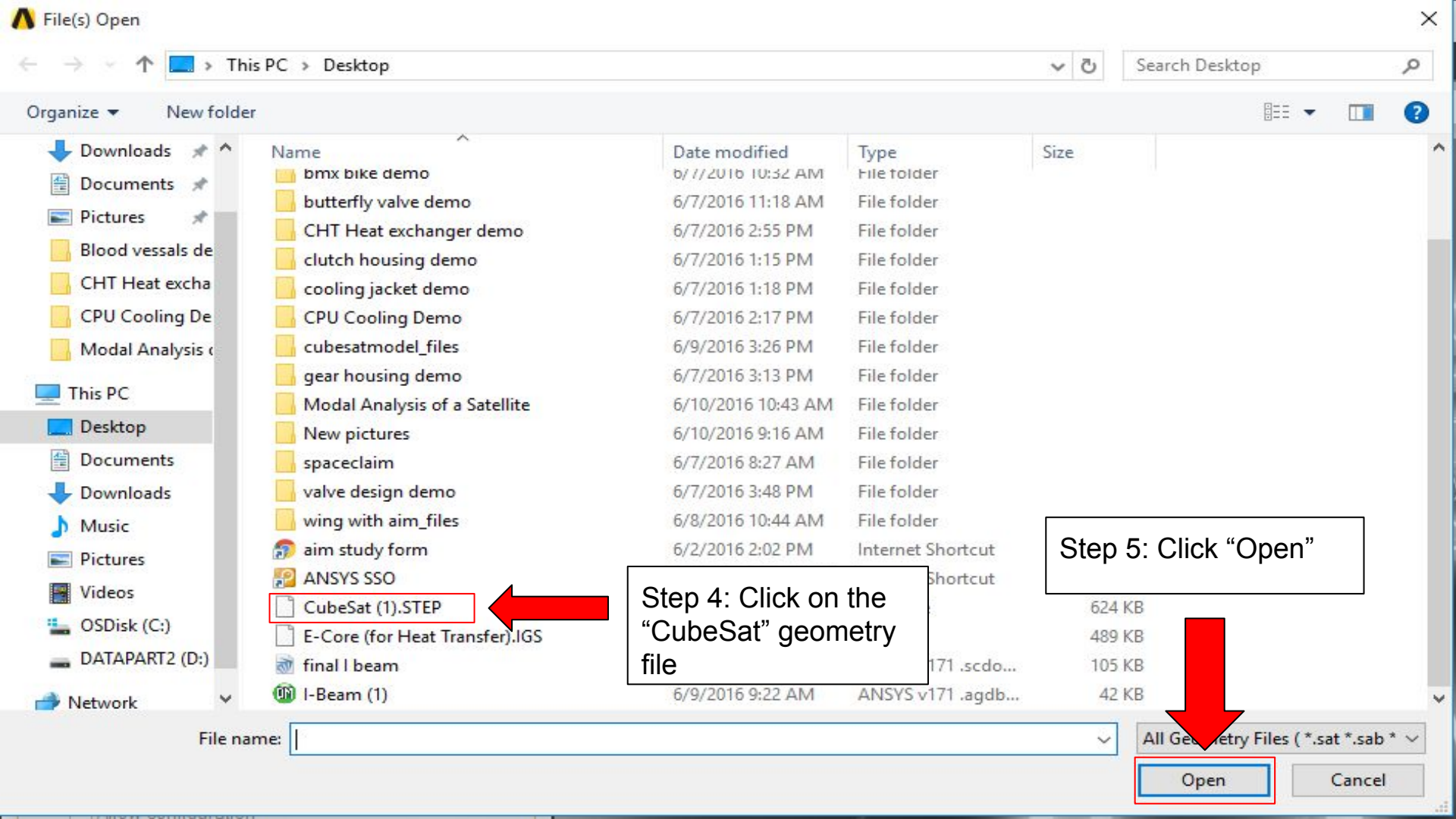
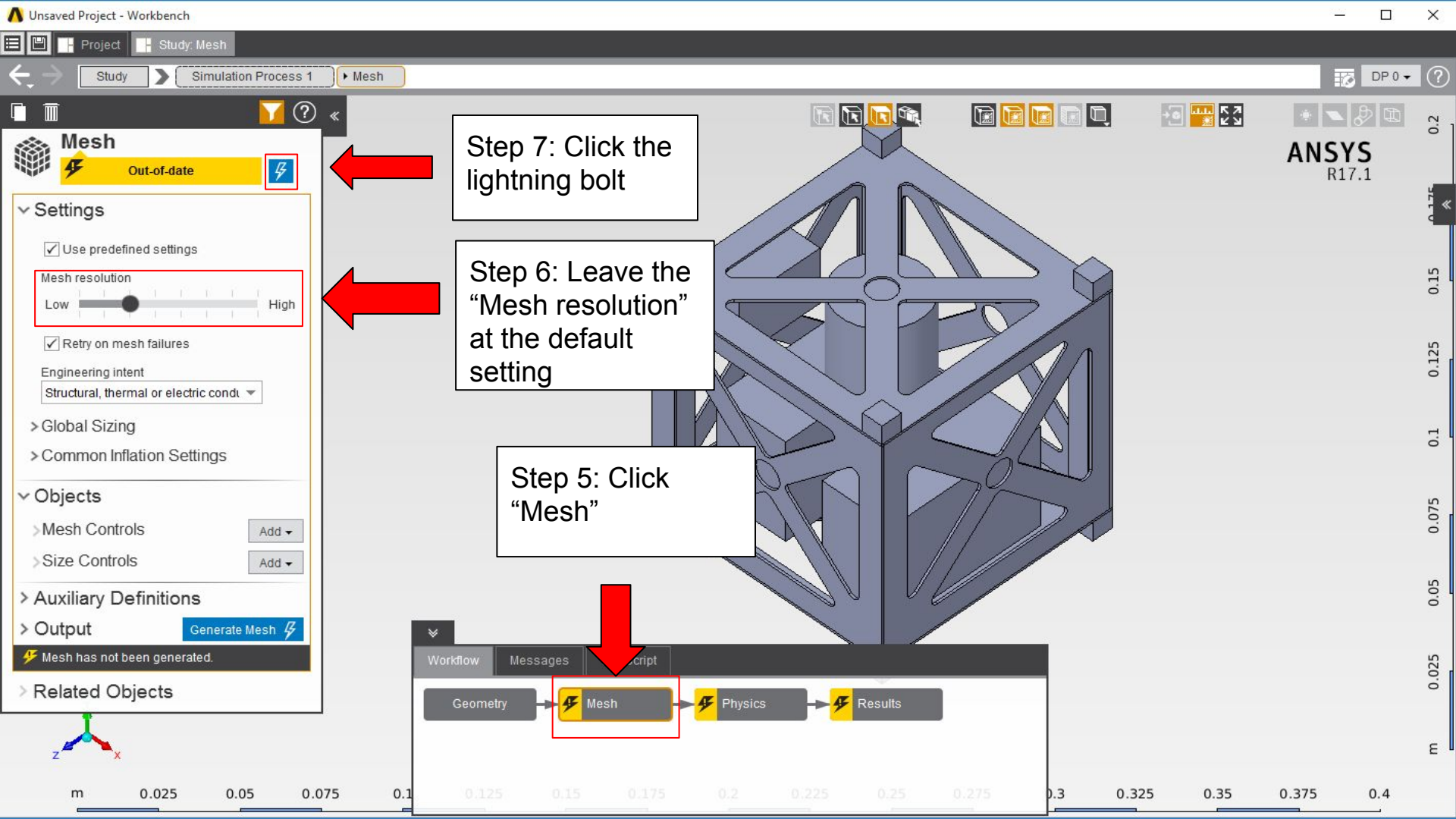
Modal Analysis of a Satellite

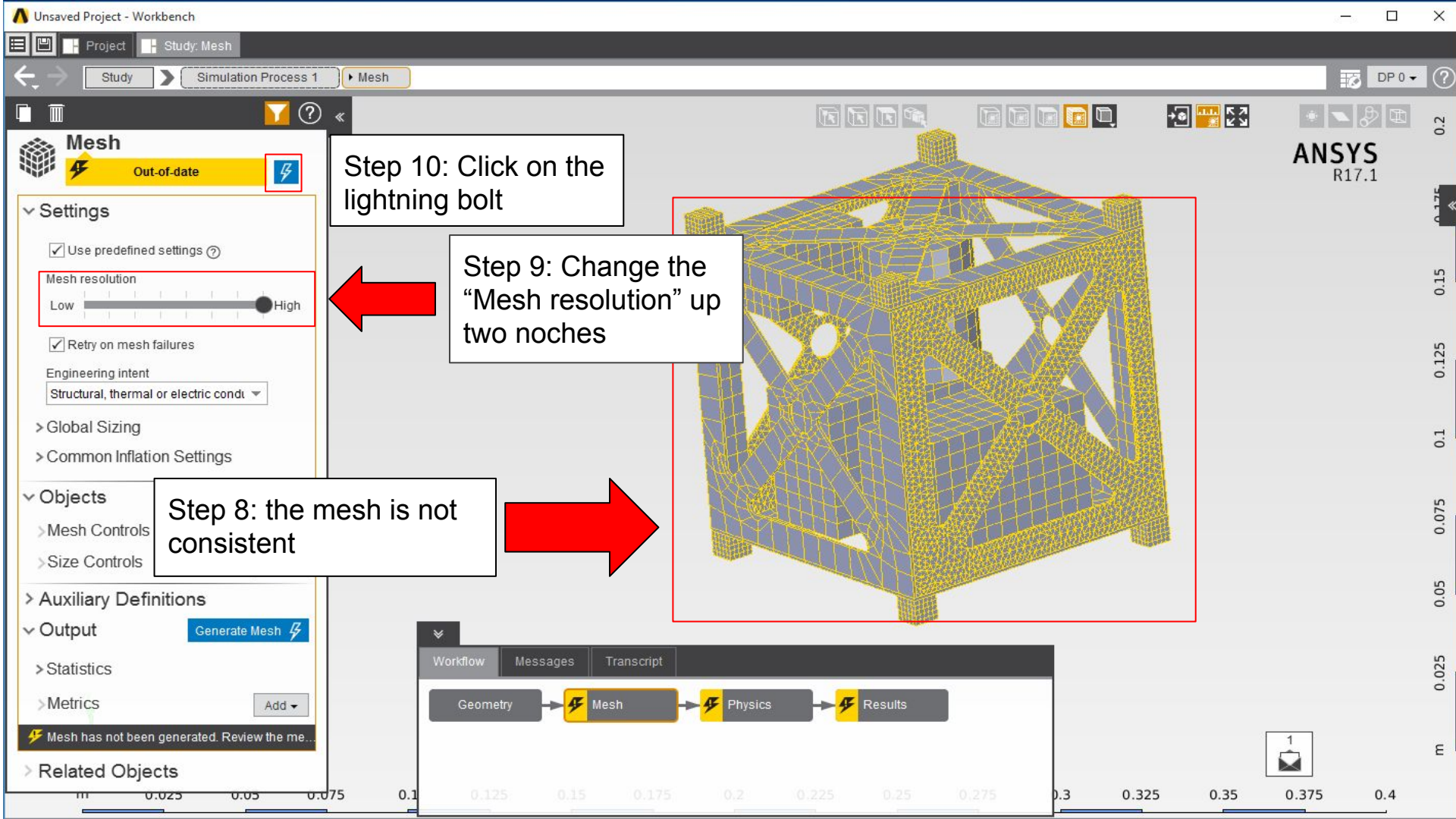
Using Ansys AIM


Unsaved Project - Workbench
Project
Study
Study
DP 0
Study
Simulation Processes
Simulation Process Templates
Structural
Fluid Flow
Conduction
Fluid-Structure Interaction
Fluid-Solid Heat Transfer
Polymer Extrusion
Magnetics
User Defined
Additional Creation Methods
Import Geometry
Connect to CAD
Import Database
Add Task
Named Expressions / Values
Step 1: Click on “Structural”
Workflow
Messages
AIM


Unsaved Project - Workbench
Project
Study: Structural Template
Study
Structural Template
DP 0
Structural Template
Geometry:
Define new geometry
Import existing geometry
Connect to active CAD session
Geometry options:
Allow model editing
Additional physics:
Thermal
Electric conduction
Calculation type:
Static
Modal
Options:
Detect contact automatically
Compute fatigue results
Allow configuration
Typical settings and results will be defined automatically.
Cancel
Create Simulation Process
Step 2: Switch calculation type to "Modal"
Step 3: Click "Create Simulation Process"
AIM


File(s) Open
This PC
Desktop
Search Desktop
Organize
New folder
Downloads
Documents
Pictures
Blood vessals de
CHT Heat excha
CPU Cooling De
Modal Analysis
This PC
Desktop
Documents
Downloads
Music
Pictures
Videos
OSDisk (C:)
DATAPART2 (D:)
Network
Name
Date modified
Type
Size
bmx bike demo
butterfly valve demo 6/7/2016 11:18 AM File folder
CHT Heat exchanger demo 6/7/2016 2:55 PM File folder
clutch housing demo 6/7/2016 1:15 PM File folder
cooling jacket demo 6/7/2016 1:18 PM File folder
CPU Cooling Demo 6/7/2016 2:17 PM File folder
cubesatmodel_files 6/9/2016 3:26 PM File folder
gear housing demo 6/7/2016 3:13 PM File folder
Modal Analysis of a Satellite 6/10/2016 10:43 AM File folder
New pictures 6/10/2016 9:16 AM File folder
spaceclaim 6/7/2016 8:27 AM File folder
valve design demo 6/7/2016 3:48 PM File folder
wing with aim_files 6/8/2016 10:44 AM File folder
aim study form 6/2/2016 2:02 PM Internet Shortcut
ANSYS SSO
CubeSat (1).STEP 624 KB
E-Core (for Heat Transfer).IGS 489 KB
final I beam 105 KB
I-Beam (1) 6/9/2016 9:22 AM ANSYS v171 .agdb... 42 KB
Step 4: Click on the “CubeSat” geometry file
Step 5: Click “Open”
File name:
All Geometry Files (*.sat *.sab *
Open
Cancel


Unsaved Project - Workbench
Project
Study: Mesh
Study
Simulation Process 1
Mesh
DP 0
Mesh
Out-of-date
Settings
Use predefined settings
Mesh resolution
Low
High
Retry on mesh failures
Engineering intent
Structural, thermal or electric condu
Global Sizing
Common Inflation Settings
Objects
Mesh Controls
Add
Size Controls
Add
Auxiliary Definitions
Output
Generate Mesh
Mesh has not been generated.
Related Objects
Step 7: Click the lightning bolt
Step 6: Leave the "Mesh resolution" at the default setting
Step 5: Click "Mesh"
Workflow
Messages
Geometry
Mesh
Physics
Results
ANSYS
R17.1
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4


Unsaved Project - Workbench
Project
Study: Mesh
Study
Simulation Process 1
Mesh
DP 0
Mesh
Out-of-date
Settings
Use predefined settings
Mesh resolution
Low
High
Retry on mesh failures
Engineering intent
Structural, thermal or electric condu
Global Sizing
Common Inflation Settings
Objects
Mesh Controls
Size Controls
Auxiliary Definitions
Output
Generate Mesh
Statistics
Metrics
Add
Mesh has not been generated. Review the me...
Related Objects
Step 10: Click on the lightning bolt
Step 9: Change the “Mesh resolution” up two noches
Step 8: the mesh is not consistent
ANSYS
R17.1
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4

Unsaved Project - Workbench

Project | Study: Mesh

Study | Simulation Process 1 | Mesh

DP 0

Mesh

Up-to-date

Settings

- Use predefined settings
- Mesh resolution: Low — High
- Retry on mesh fai
- Engineering intent: Structural, thermal or electric condu
- Global Sizing
- Common Inflation Settings

Objects

- Mesh Controls — Add
- Size Controls — Add

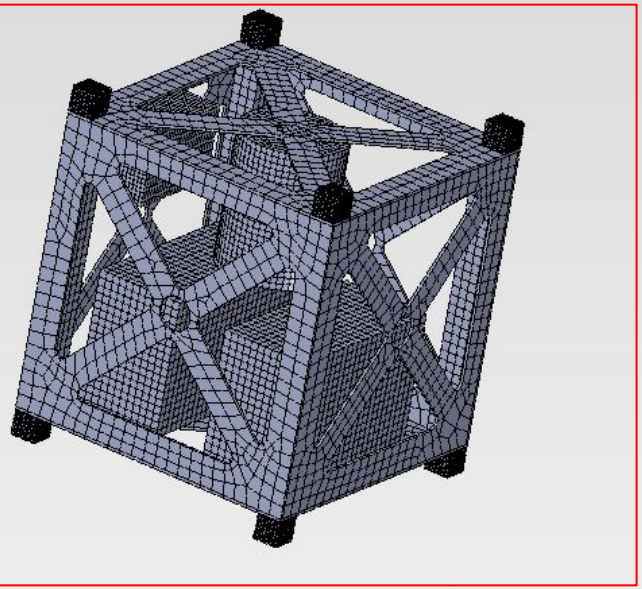
Auxiliary Definitions

Output — Generate Mesh

- Statistics
- Metrics — Add

Related Objects

Mesh looks way better



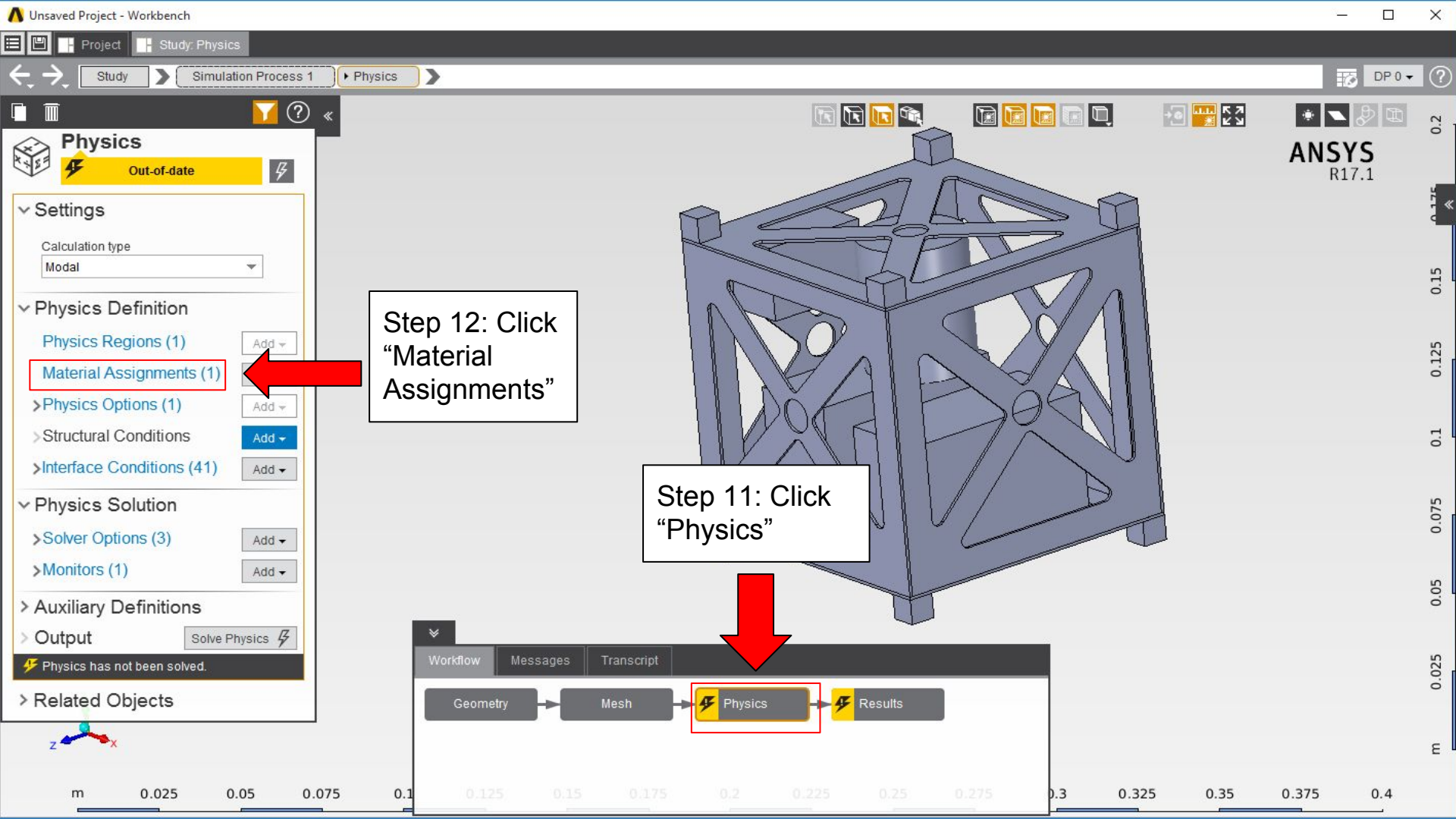
ANSYS
R17.1

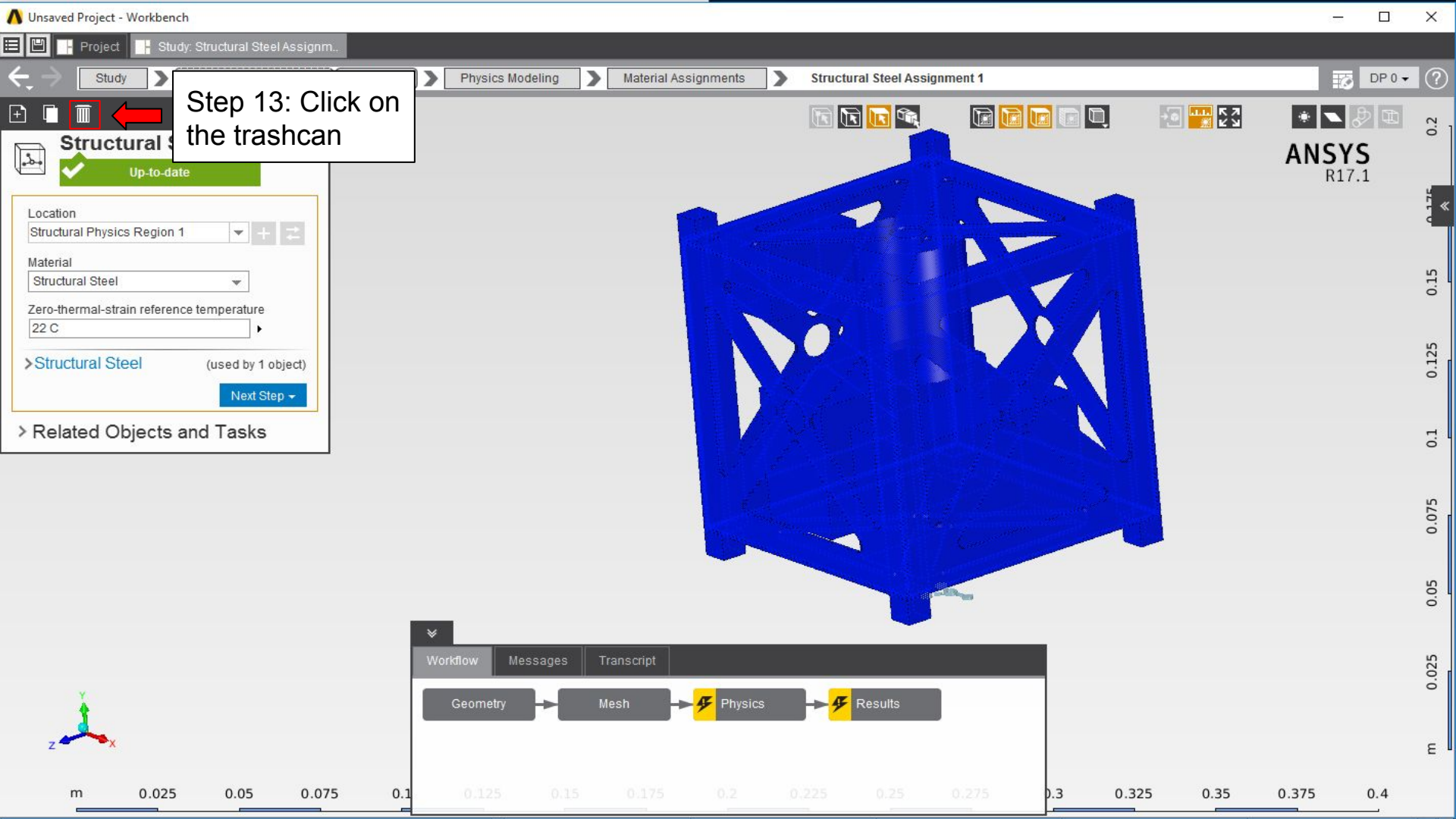
Workflow | Messages | Transcript

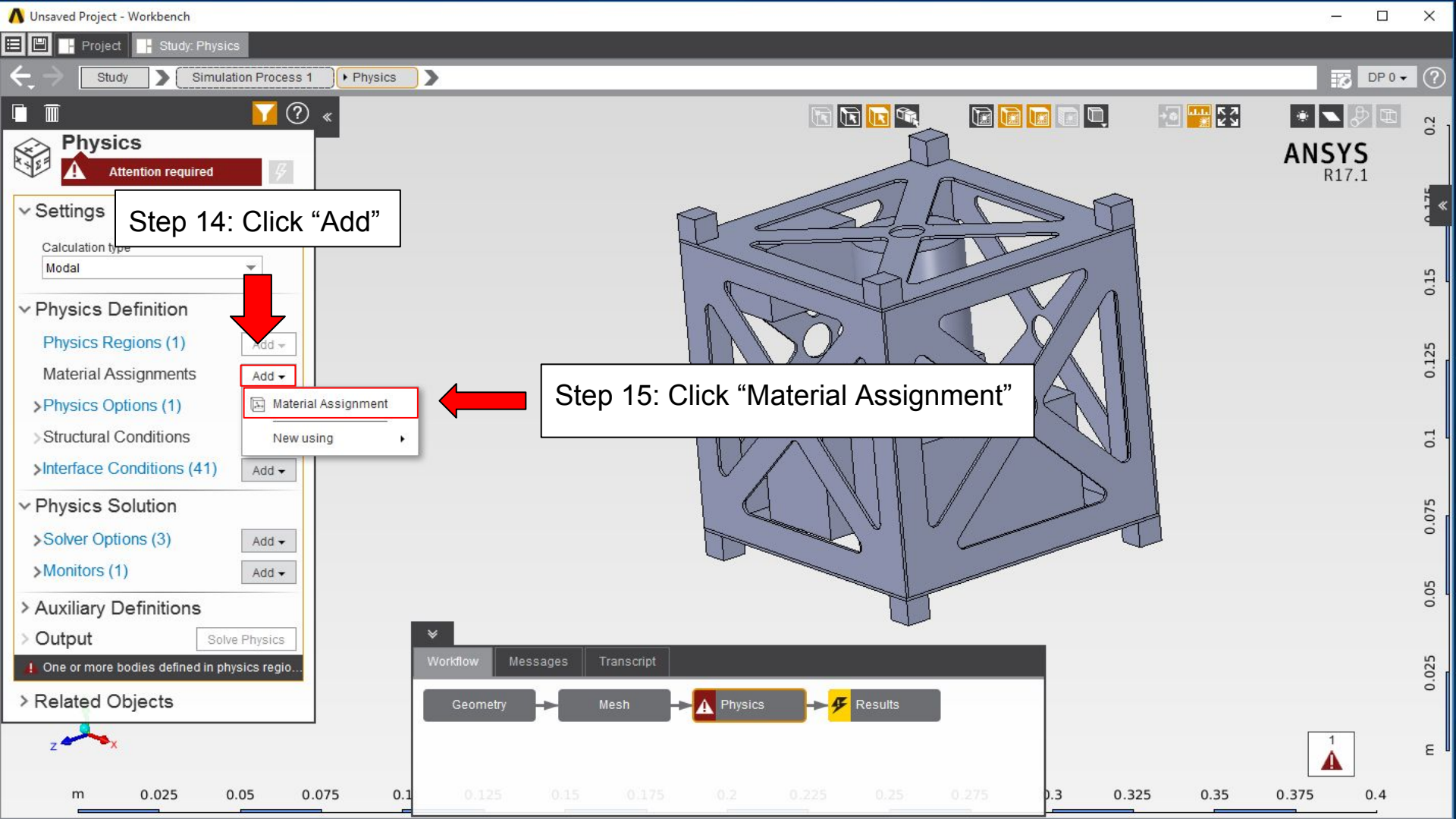
Geometry → Mesh → Physics → Results

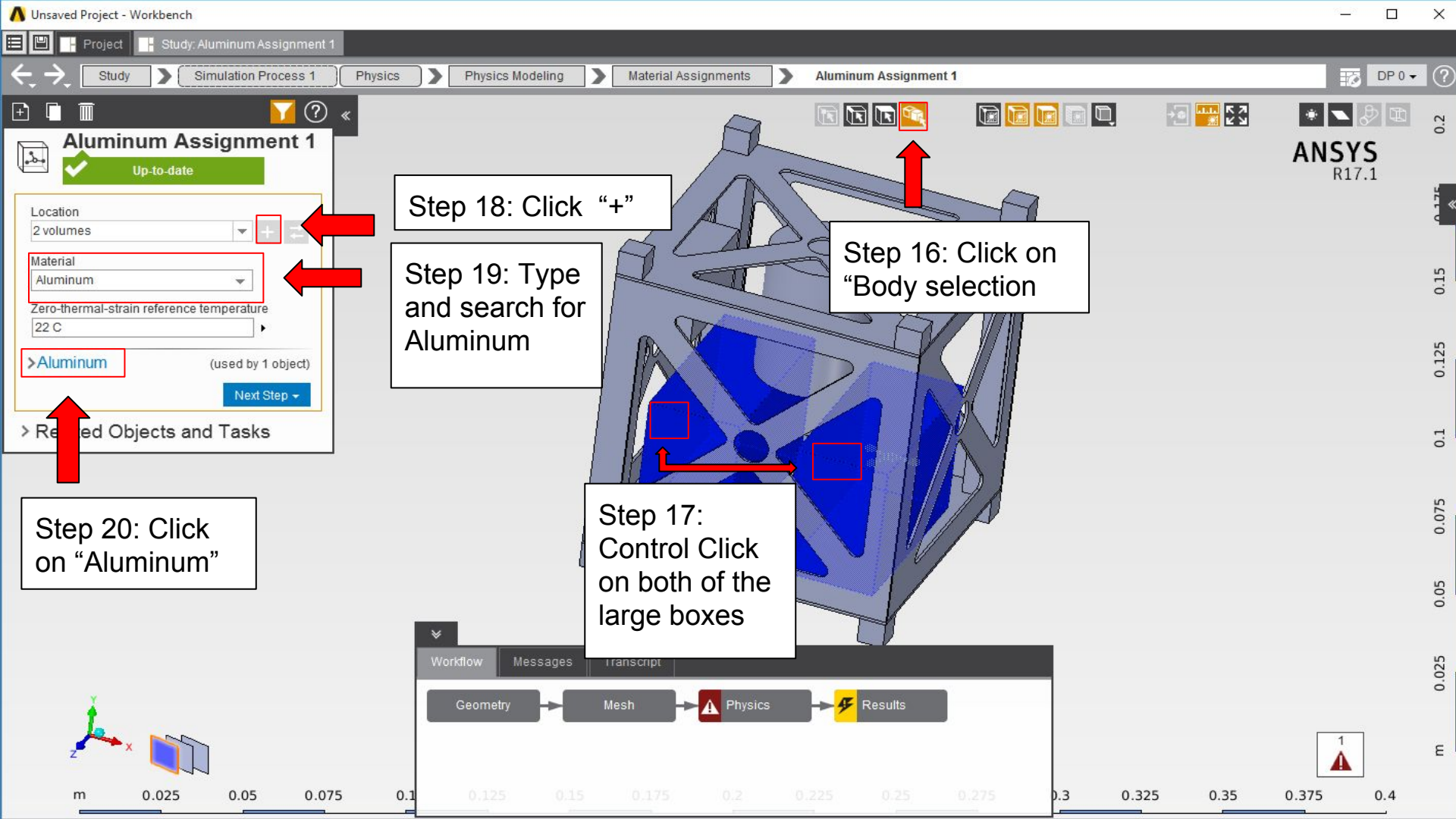
0.25 0.2 0.15 0.1 0.05 m

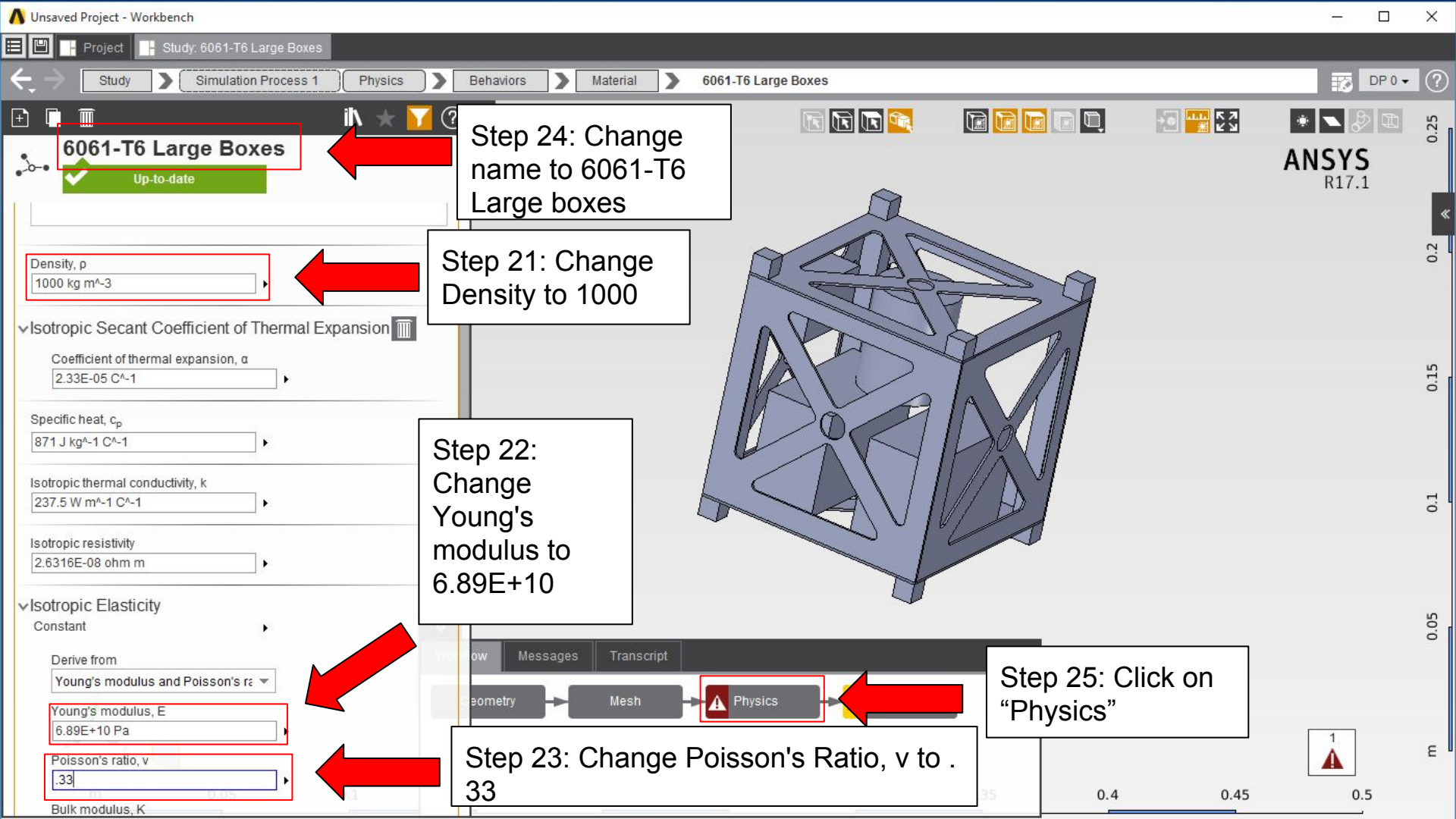
0.1 0.15 0.2 0.25 0.3 0.35 0.4 0.45 0.5

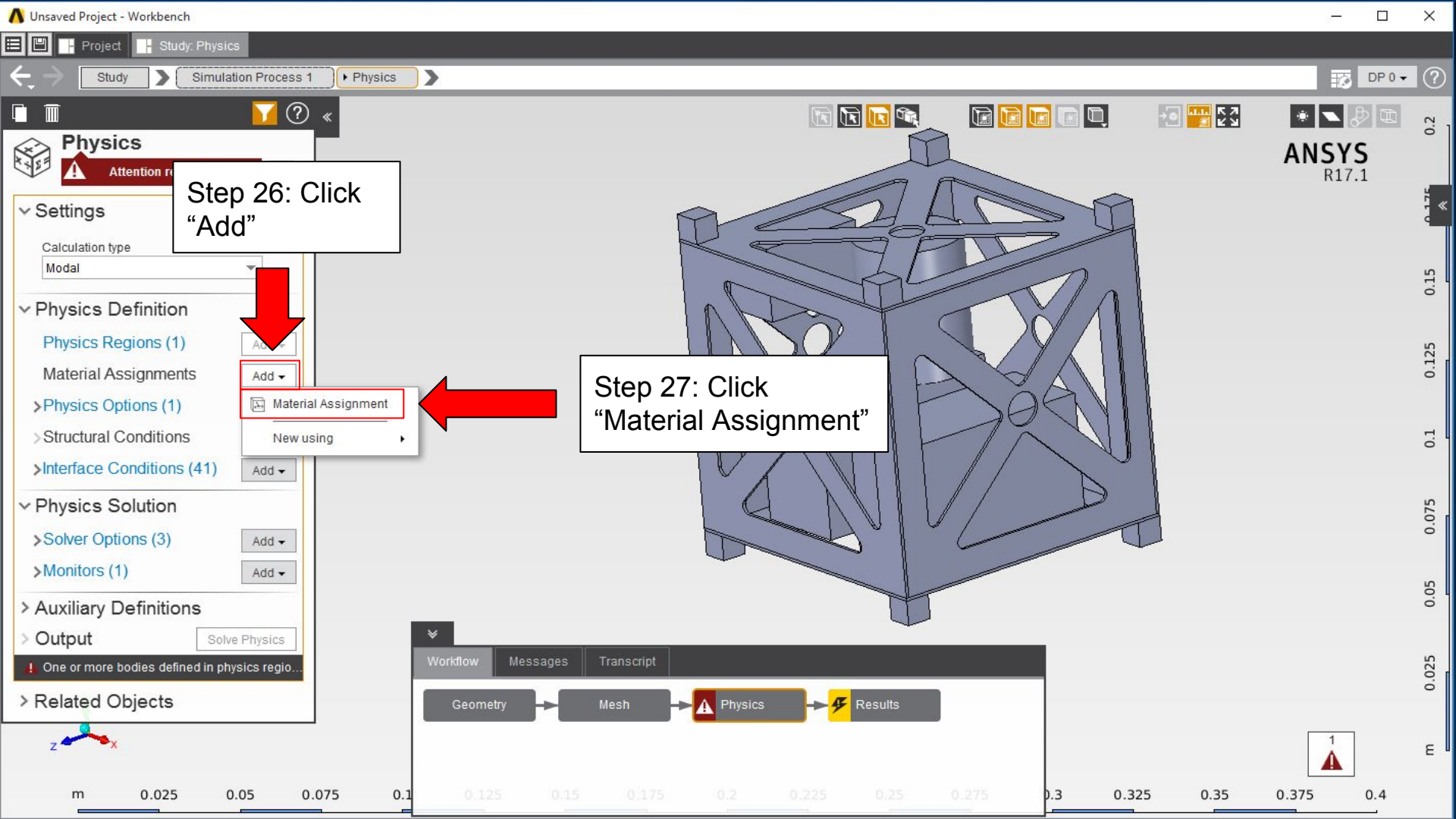

Unsaved Project - Workbench
Project
Study: Physics
Study
Simulation Process 1
Physics
DP 0
Physics
Out-of-date
Settings
Calculation type
Modal
Physics Definition
Physics Regions (1)
Add
Material Assignments (1)
Physics Options (1)
Add
Structural Conditions
Add
Interface Conditions (41)
Add
Physics Solution
Solver Options (3)
Add
Monitors (1)
Add
Auxiliary Definitions
Output
Solve Physics
Physics has not been solved.
Related Objects
Step 12: Click "Material Assignments"
Step 11: Click "Physics"
ANSYS
R17.1
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4

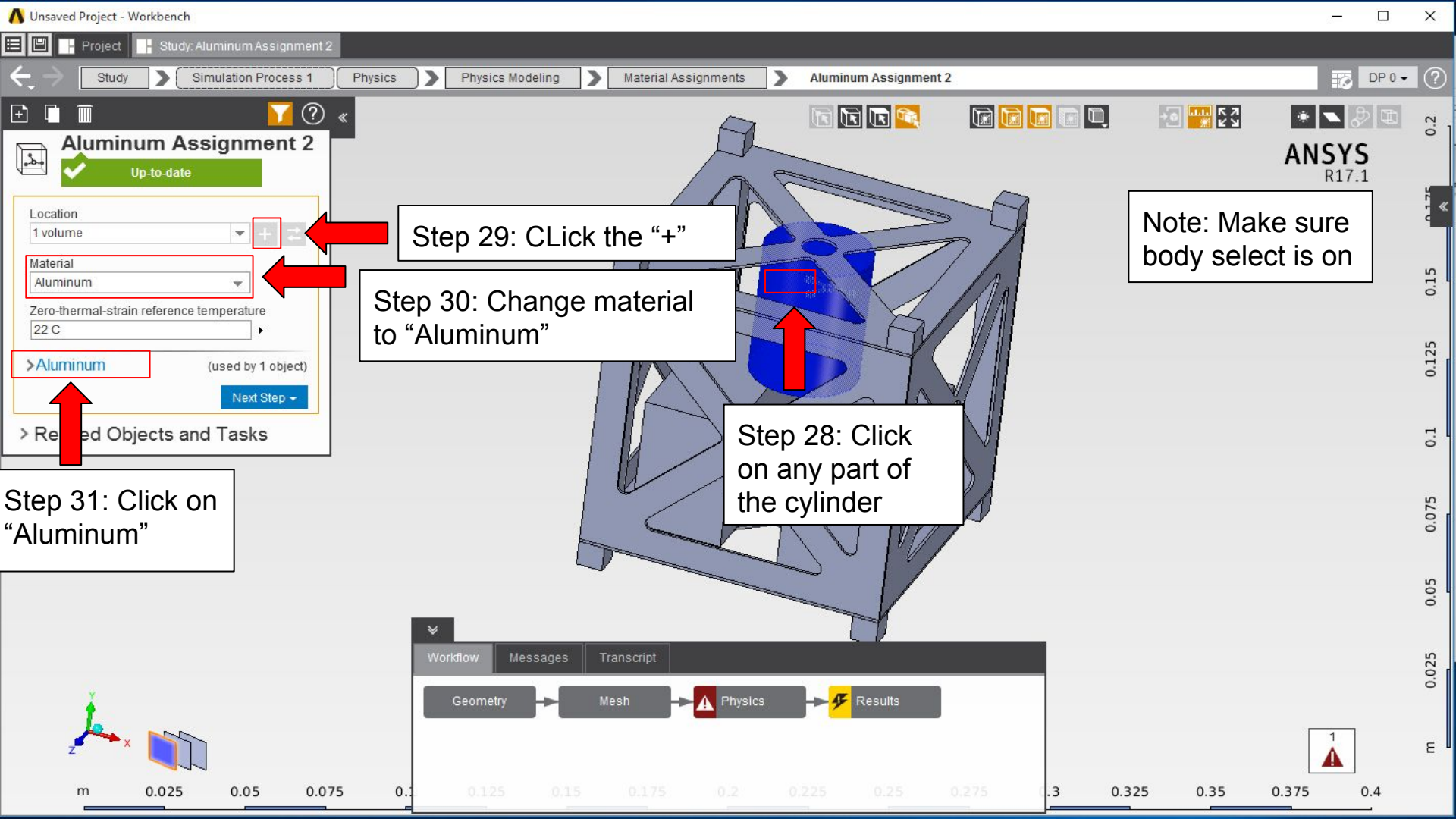

Unsaved Project - Workbench
Project
Study: Structural Steel Assignm...
Study
Physics Modeling
Material Assignments
Structural Steel Assignment 1
DP 0
Step 13: Click on the trashcan
Structural S
Up-to-date
Location
Structural Physics Region 1
Material
Structural Steel
Zero-thermal-strain reference temperature
22 C
Structural Steel
(used by 1 object)
Next Step
Related Objects and Tasks
ANSYS
R17.1
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4

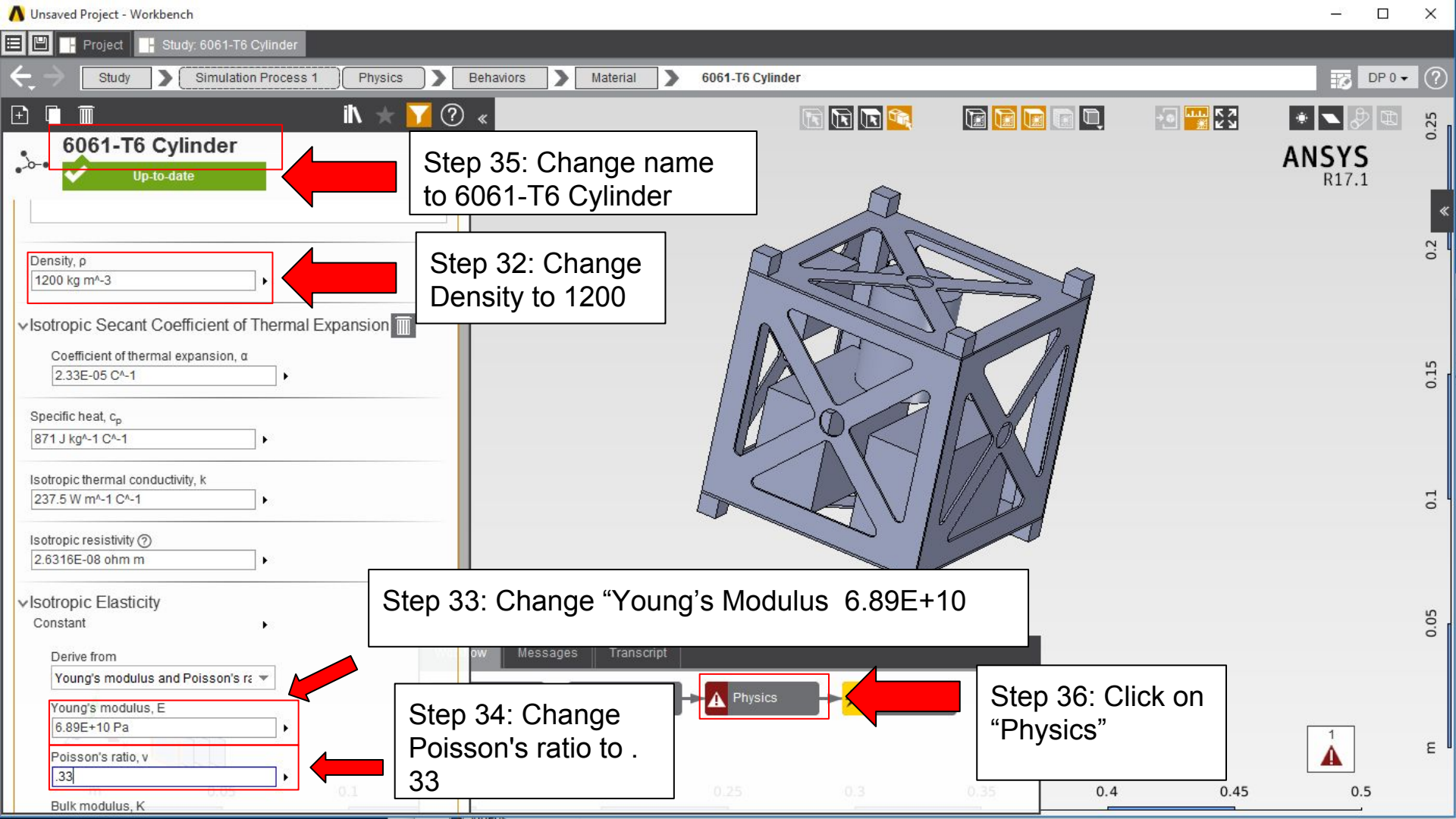

Unsaved Project - Workbench
Project
Study: Physics
Study
Simulation Process 1
Physics
DP 0
Physics
Attention required
Settings
Calculation type
Modal
Physics Definition
Physics Regions (1)
Add
Material Assignments
Add
Physics Options (1)
Structural Conditions
Interface Conditions (41)
Add
Material Assignment
New using
Physics Solution
Solver Options (3)
Add
Monitors (1)
Add
Auxiliary Definitions
Output
Solve Physics
One or more bodies defined in physics regio...
Related Objects
Step 14: Click “Add”
Step 15: Click “Material Assignment”
ANSYS
R17.1
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4

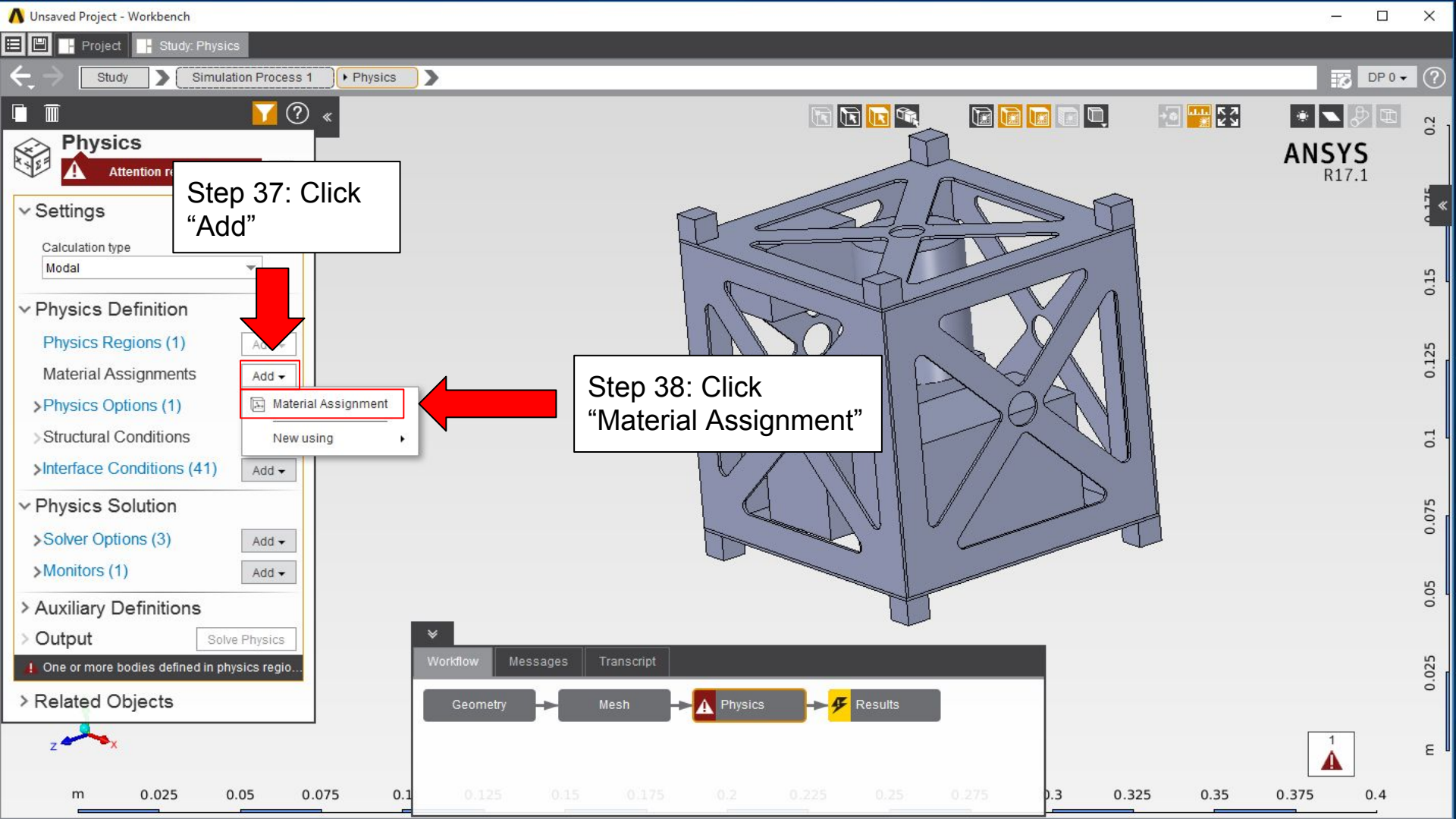

Unsaved Project - Workbench
Project
Study: Aluminum Assignment 1
Study
Simulation Process 1
Physics
Physics Modeling
Material Assignments
Aluminum Assignment 1
DP 0
Aluminum Assignment 1
Up-to-date
Location
2 volumes
Material
Aluminum
Zero-thermal-strain reference temperature
22 C
>Aluminum
(used by 1 object)
Next Step
> Related Objects and Tasks
Step 18: Click "+"
Step 19: Type and search for Aluminum
Step 16: Click on "Body selection
Step 17: Control Click on both of the large boxes
Step 20: Click on "Aluminum"
ANSYS
R17.1
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4

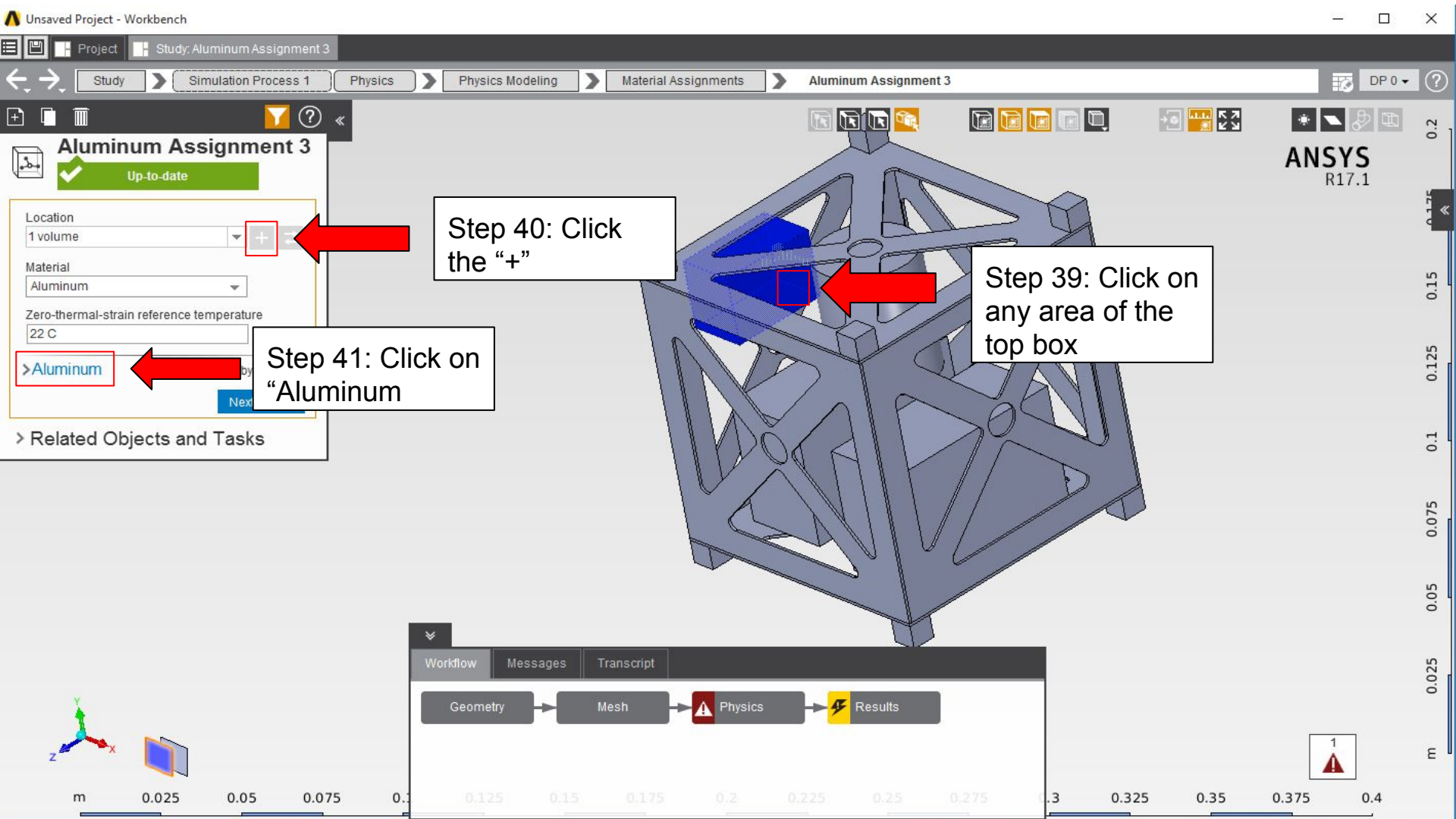

Unsaved Project - Workbench
Project
Study: 6061-T6 Large Boxes
Study
Simulation Process 1
Physics
Behaviors
Material
6061-T6 Large Boxes
DP 0
6061-T6 Large Boxes
Up-to-date
Step 24: Change name to 6061-T6 Large boxes
Density, ρ
1000 kg m^-3
Step 21: Change Density to 1000
Isotropic Secant Coefficient of Thermal Expansion
Coefficient of thermal expansion, α
2.33E-05 C^-1
Specific heat, c_p
871 J kg^-1 C^-1
Isotropic thermal conductivity, k
237.5 W m^-1 C^-1
Isotropic resistivity
2.6316E-08 ohm m
Step 22: Change Young's modulus to 6.89E+10
Isotropic Elasticity
Constant
Derive from
Young's modulus and Poisson's ra
Young's modulus, E
6.89E+10 Pa
Poisson's ratio, v
.33
Bulk modulus, K
Step 23: Change Poisson's Ratio, v to . 33
ANSYS
R17.1
Messages
Transcript
Mesh
Physics
Step 25: Click on "Physics"
0.25
0.2
0.15
0.1
0.05
m
0.4
0.45
0.5

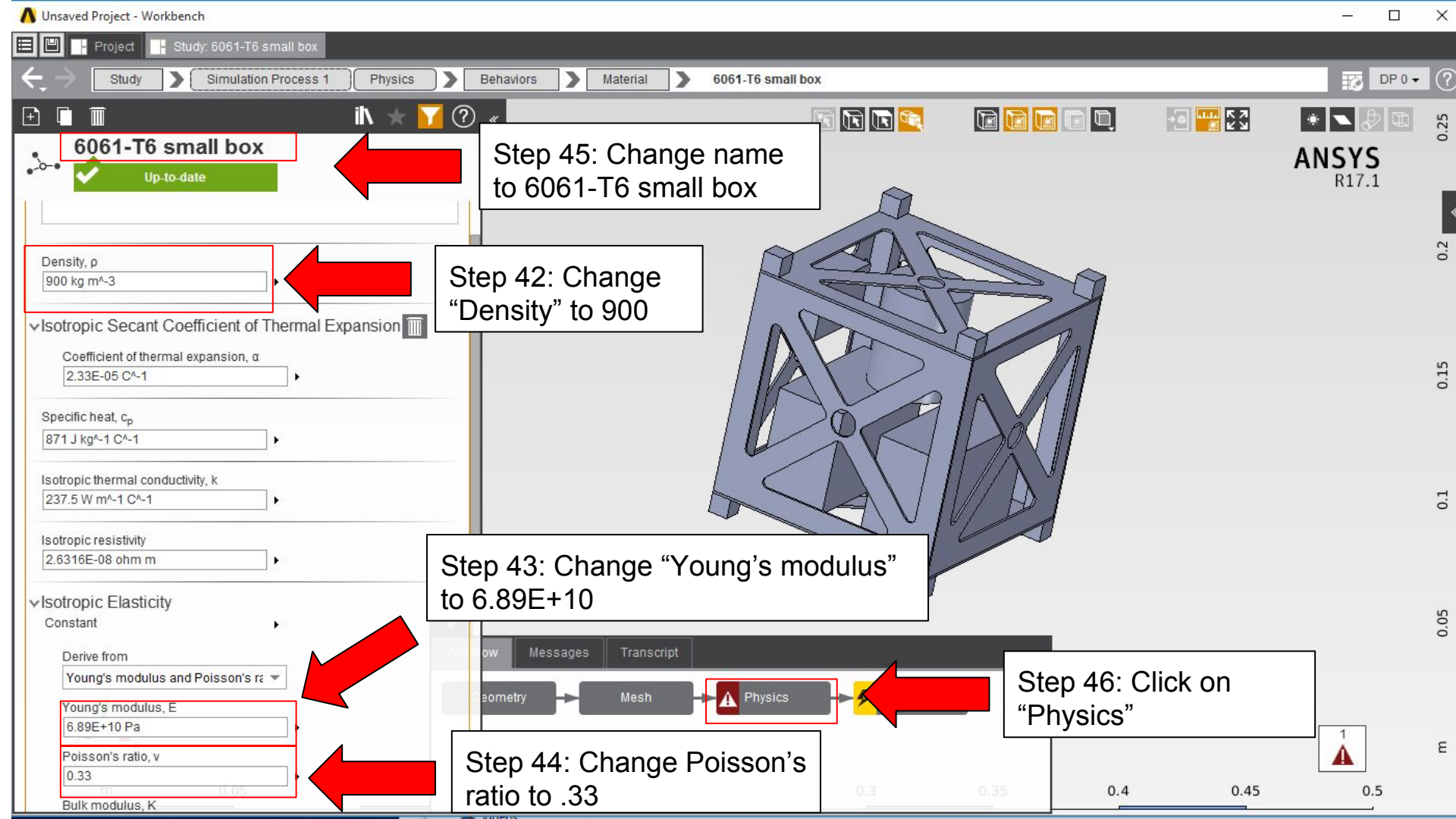

Unsaved Project - Workbench
Project
Study: Physics
Study
Simulation Process 1
Physics
DP 0
Physics
Attention r
Step 26: Click "Add"
Settings
Calculation type
Modal
Physics Definition
Physics Regions (1)
Material Assignments
Add
Material Assignment
New using
Physics Options (1)
Structural Conditions
Interface Conditions (41)
Physics Solution
Solver Options (3)
Monitors (1)
Auxiliary Definitions
Output
Solve Physics
One or more bodies defined in physics regio...
Related Objects
Step 27: Click "Material Assignment"
ANSYS
R17.1
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4

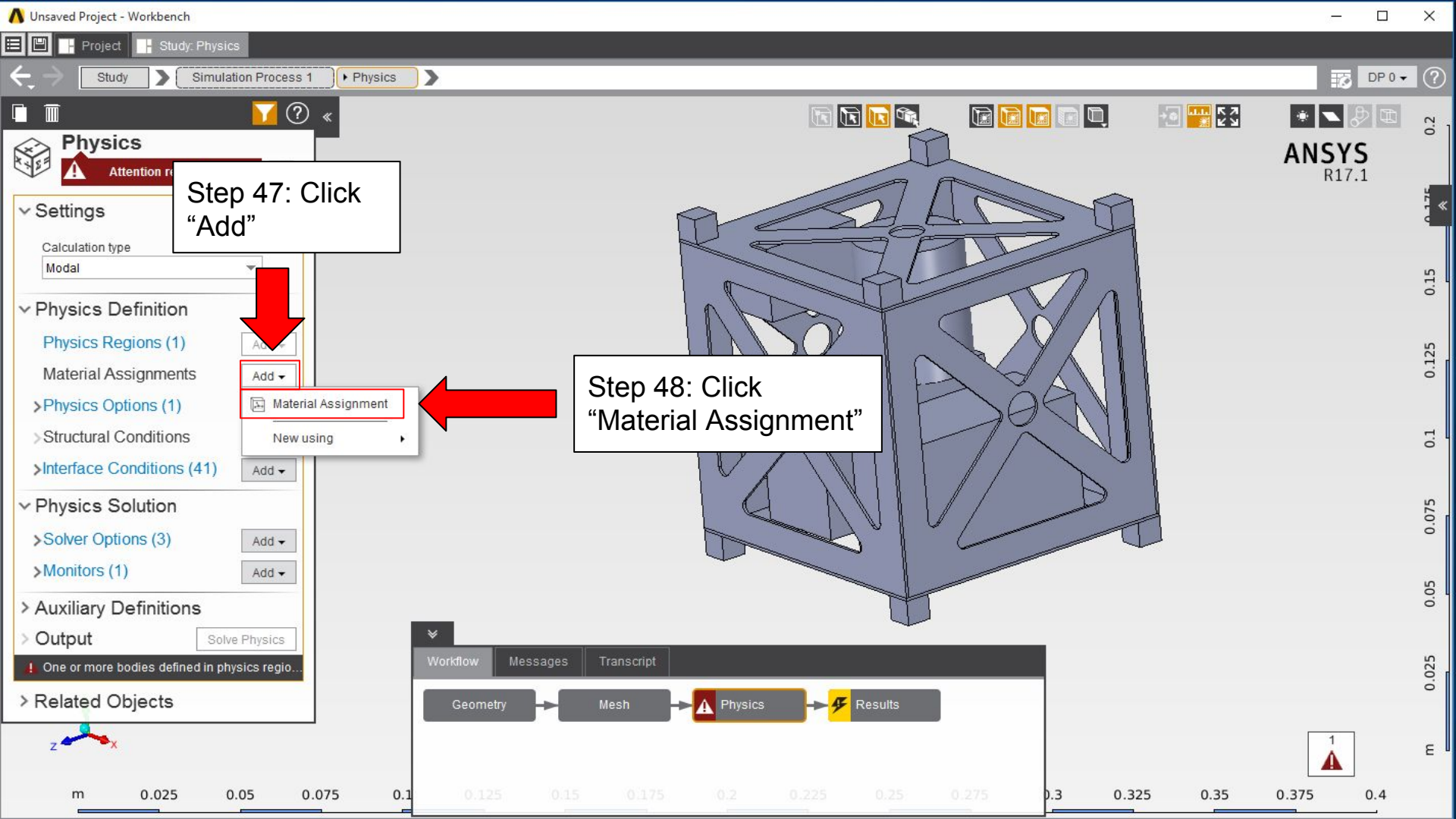

Unsaved Project - Workbench
Project
Study: Aluminum Assignment 2
Study
Simulation Process 1
Physics
Physics Modeling
Material Assignments
Aluminum Assignment 2
DP 0
Aluminum Assignment 2
Up-to-date
Location
1 volume
Material
Aluminum
Zero-thermal-strain reference temperature
22 C
>Aluminum
(used by 1 object)
Next Step
> Related Objects and Tasks
Step 29: CLick the “+”
Step 30: Change material to “Aluminum”
Step 31: Click on “Aluminum”
Step 28: Click on any part of the cylinder
Note: Make sure body select is on
ANSYS
R17.1
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.325
0.35
0.375
0.4

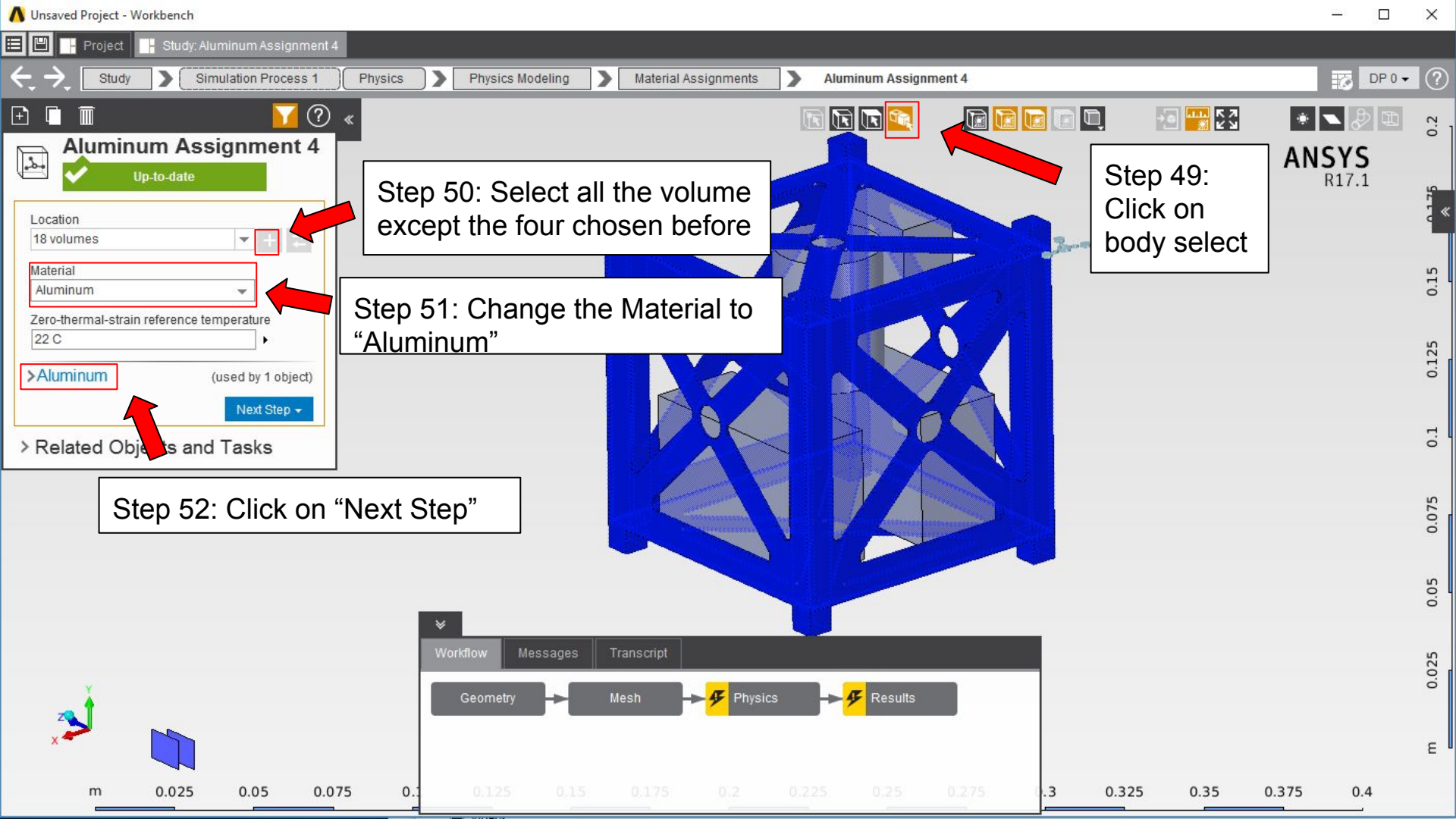

Unsaved Project - Workbench
Project
Study: 6061-T6 Cylinder
Study
Simulation Process 1
Physics
Behaviors
Material
6061-T6 Cylinder
DP 0
6061-T6 Cylinder
Up-to-date
Step 35: Change name to 6061-T6 Cylinder
Density, ρ
1200 kg m^-3
Step 32: Change Density to 1200
Isotropic Secant Coefficient of Thermal Expansion
Coefficient of thermal expansion, α
2.33E-05 C^-1
Specific heat, cp
871 J kg^-1 C^-1
Isotropic thermal conductivity, k
237.5 W m^-1 C^-1
Isotropic resistivity
2.6316E-08 ohm m
Isotropic Elasticity
Constant
Derive from
Young's modulus and Poisson's ra
Young's modulus, E
6.89E+10 Pa
Poisson's ratio, ν
.33
Bulk modulus, K
Step 33: Change “Young’s Modulus 6.89E+10
Step 34: Change Poisson's ratio to . 33
ANSYS
R17.1
Messages
Transcript
Physics
Step 36: Click on “Physics”
0.25
0.2
0.15
0.1
0.05
m
0.4
0.45
0.5

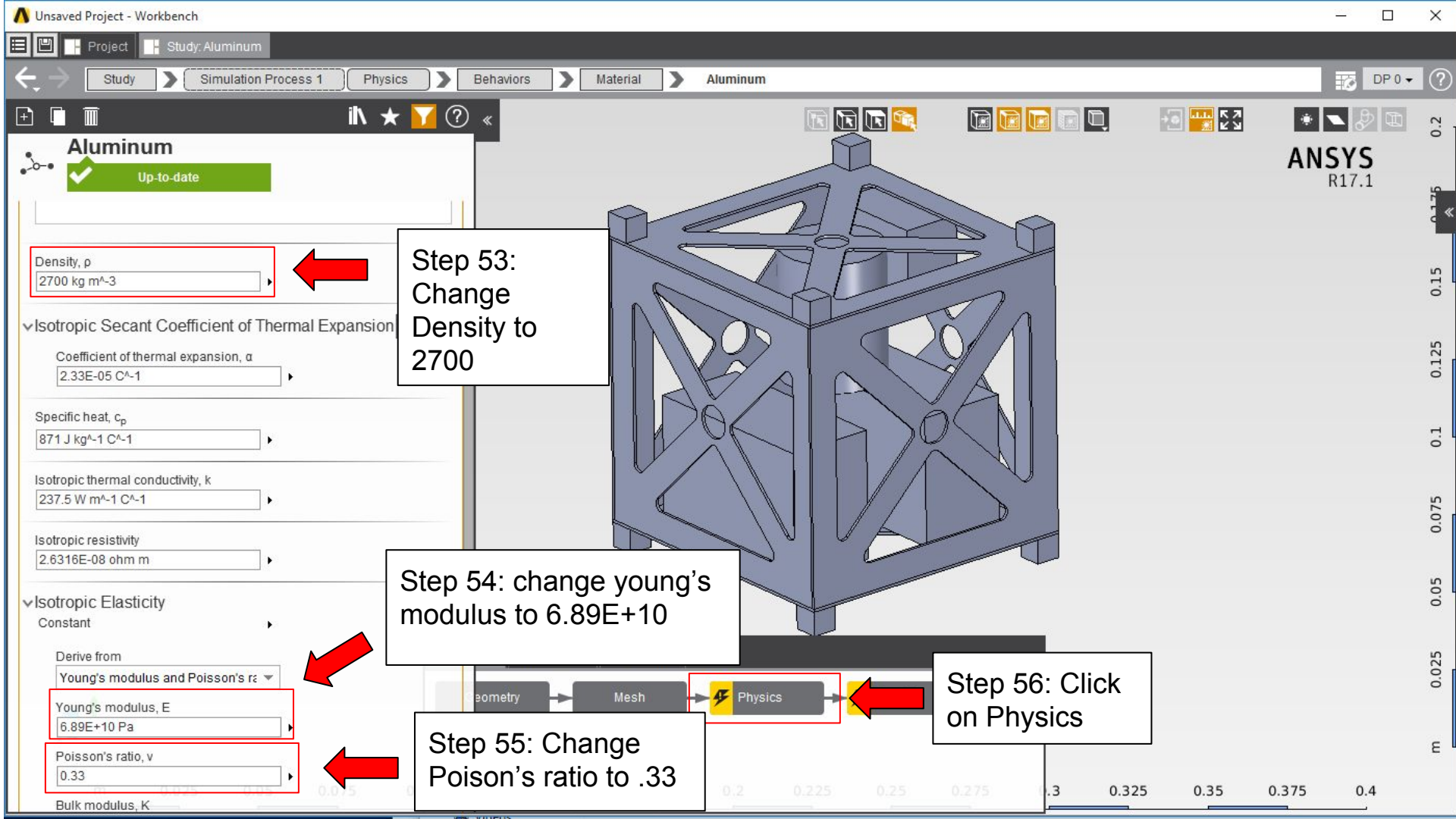

Unsaved Project - Workbench
Project
Study: Physics
Study
Simulation Process 1
Physics
DP 0
Physics
Attention r
Settings
Calculation type
Modal
Physics Definition
Physics Regions (1)
Material Assignments
Add
Material Assignment
New using
Physics Options (1)
Structural Conditions
Interface Conditions (41)
Physics Solution
Solver Options (3)
Monitors (1)
Auxiliary Definitions
Output
Solve Physics
One or more bodies defined in physics regio...
Related Objects
Step 37: Click "Add"
Step 38: Click "Material Assignment"
ANSYS
R17.1
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4

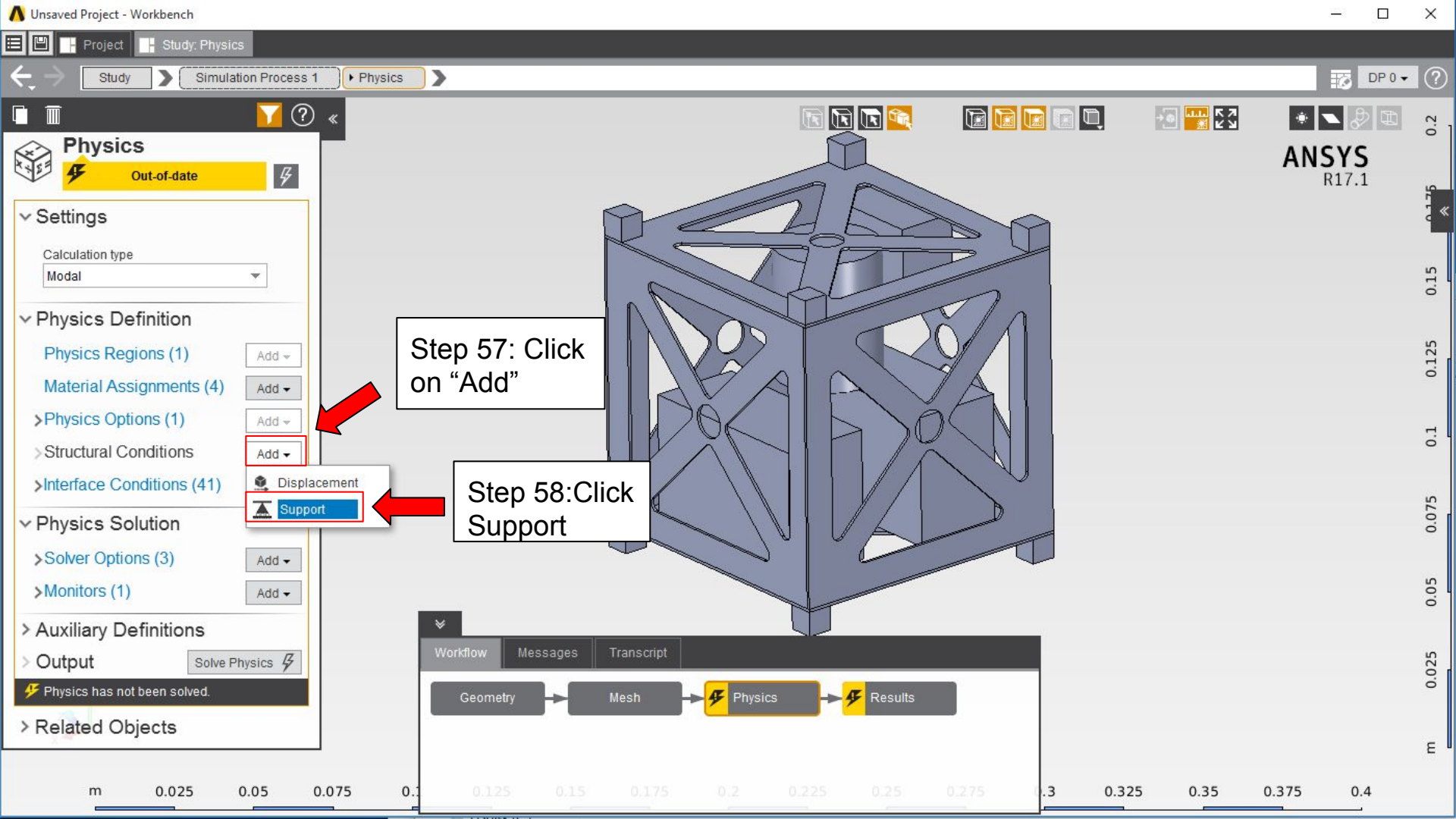

Unsaved Project - Workbench
Project
Study: Aluminum Assignment 3
Study
Simulation Process 1
Physics
Physics Modeling
Material Assignments
Aluminum Assignment 3
DP 0
Aluminum Assignment 3
Up-to-date
Location
1 volume
Material
Aluminum
Zero-thermal-strain reference temperature
22 C
>Aluminum
Next
> Related Objects and Tasks
Step 40: Click the “+”
Step 39: Click on any area of the top box
Step 41: Click on “Aluminum
ANSYS
R17.1
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4

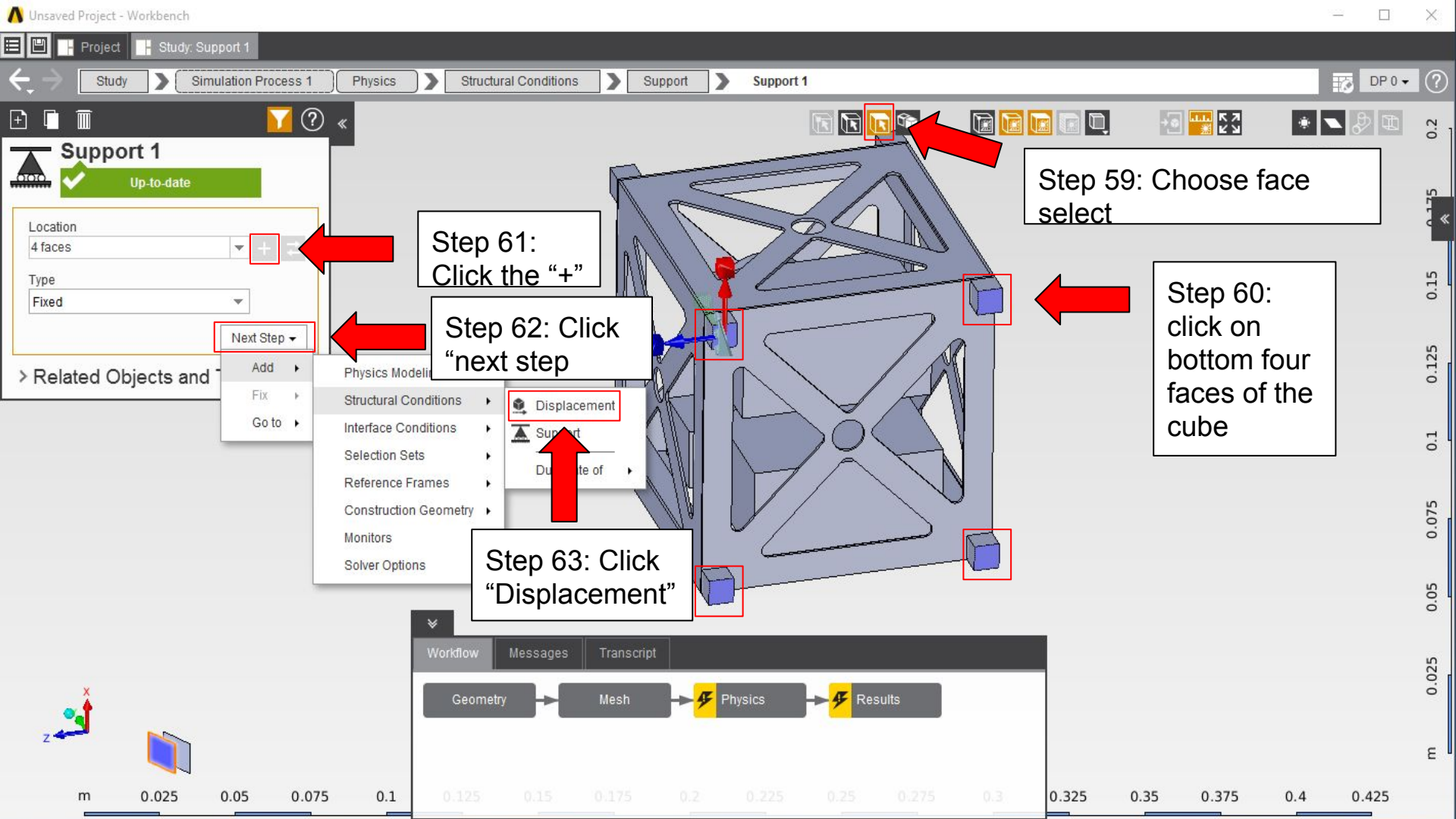

Unsaved Project - Workbench
Project
Study: 6061-T6 small box
Study
Simulation Process 1
Physics
Behaviors
Material
6061-T6 small box
DP 0
6061-T6 small box
Up-to-date
Step 45: Change name to 6061-T6 small box
Density, ρ
900 kg m^-3
Step 42: Change "Density" to 900
Isotropic Secant Coefficient of Thermal Expansion
Coefficient of thermal expansion, α
2.33E-05 C^-1
Specific heat, c_p
871 J kg^-1 C^-1
Isotropic thermal conductivity, k
237.5 W m^-1 C^-1
Isotropic resistivity
2.6316E-08 ohm m
Step 43: Change "Young's modulus" to 6.89E+10
Isotropic Elasticity
Constant
Derive from
Young's modulus and Poisson's ra
Young's modulus, E
6.89E+10 Pa
Poisson's ratio, v
0.33
Bulk modulus, K
Step 44: Change Poisson's ratio to .33
ANSYS
R17.1
Messages
Transcript
Mesh
Physics
Step 46: Click on "Physics"
0.4
0.45
0.5
m

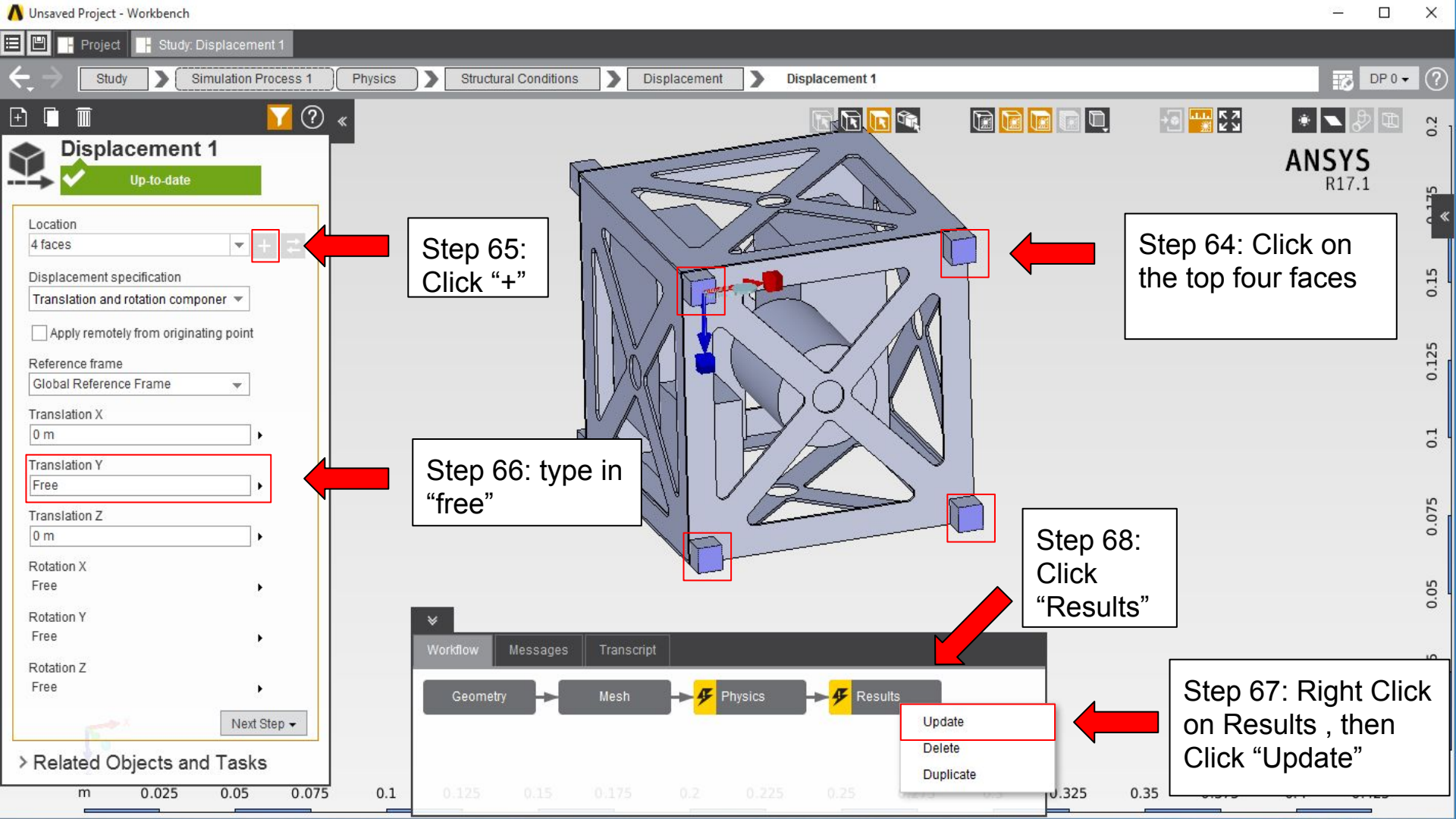

Unsaved Project - Workbench
Project
Study: Physics
Study
Simulation Process 1
Physics
DP 0
Physics
Attention r
Step 47: Click "Add"
Settings
Calculation type
Modal
Physics Definition
Physics Regions (1)
Material Assignments
Add
Material Assignment
New using
Physics Options (1)
Structural Conditions
Interface Conditions (41)
Physics Solution
Solver Options (3)
Monitors (1)
Auxiliary Definitions
Output
Solve Physics
One or more bodies defined in physics regio...
Related Objects
Step 48: Click "Material Assignment"
ANSYS
R17.1
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4

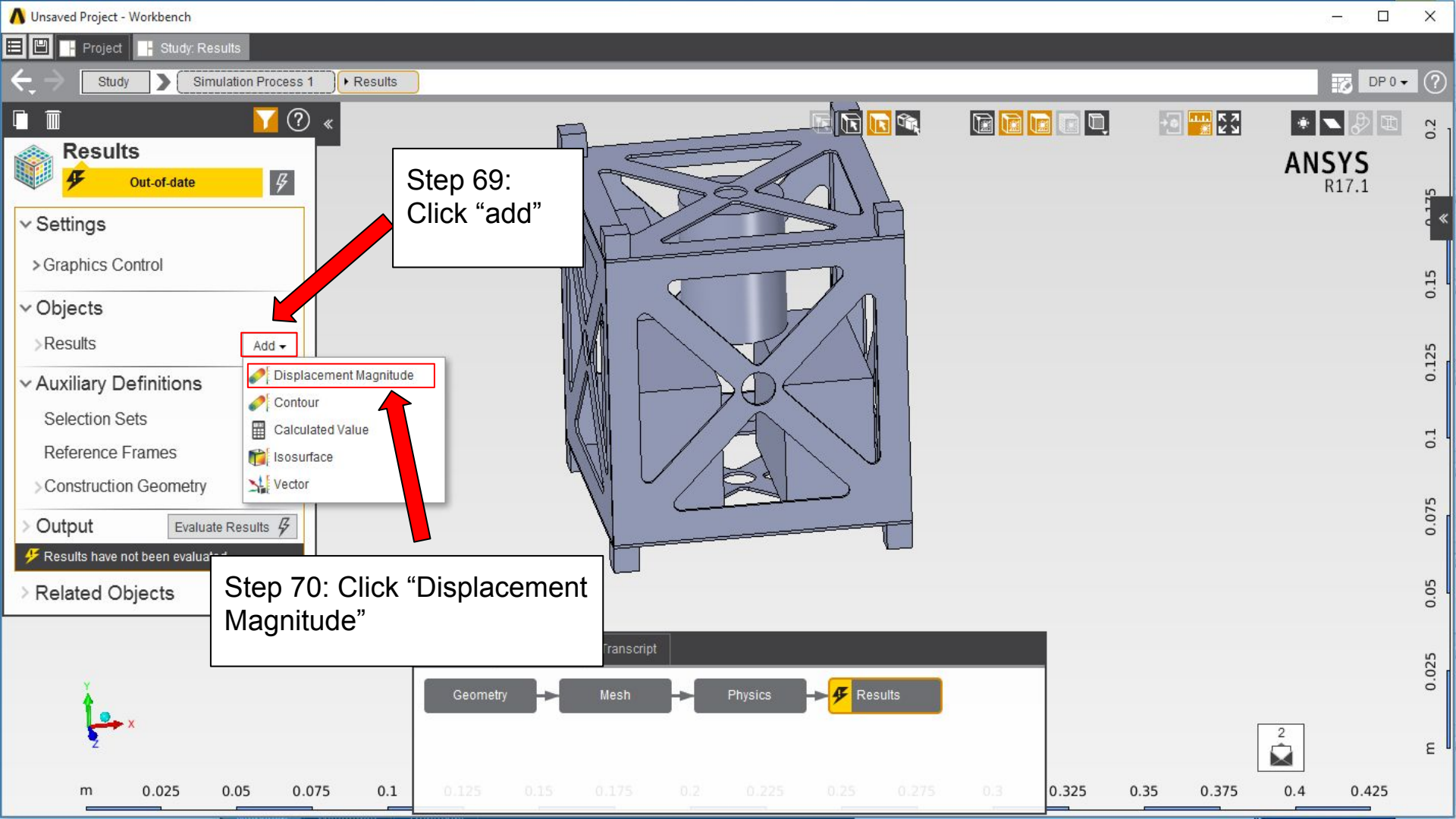

Unsaved Project - Workbench
Project
Study: Aluminum Assignment 4
Study
Simulation Process 1
Physics
Physics Modeling
Material Assignments
Aluminum Assignment 4
DP 0
Aluminum Assignment 4
Up-to-date
Location
18 volumes
Material
Aluminum
Zero-thermal-strain reference temperature
22 C
>Aluminum
(used by 1 object)
Next Step
> Related Objects and Tasks
Step 49: Click on body select
Step 50: Select all the volume except the four chosen before
Step 51: Change the Material to “Aluminum”
Step 52: Click on “Next Step”
ANSYS
R17.1
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.325
0.35
0.375
0.4

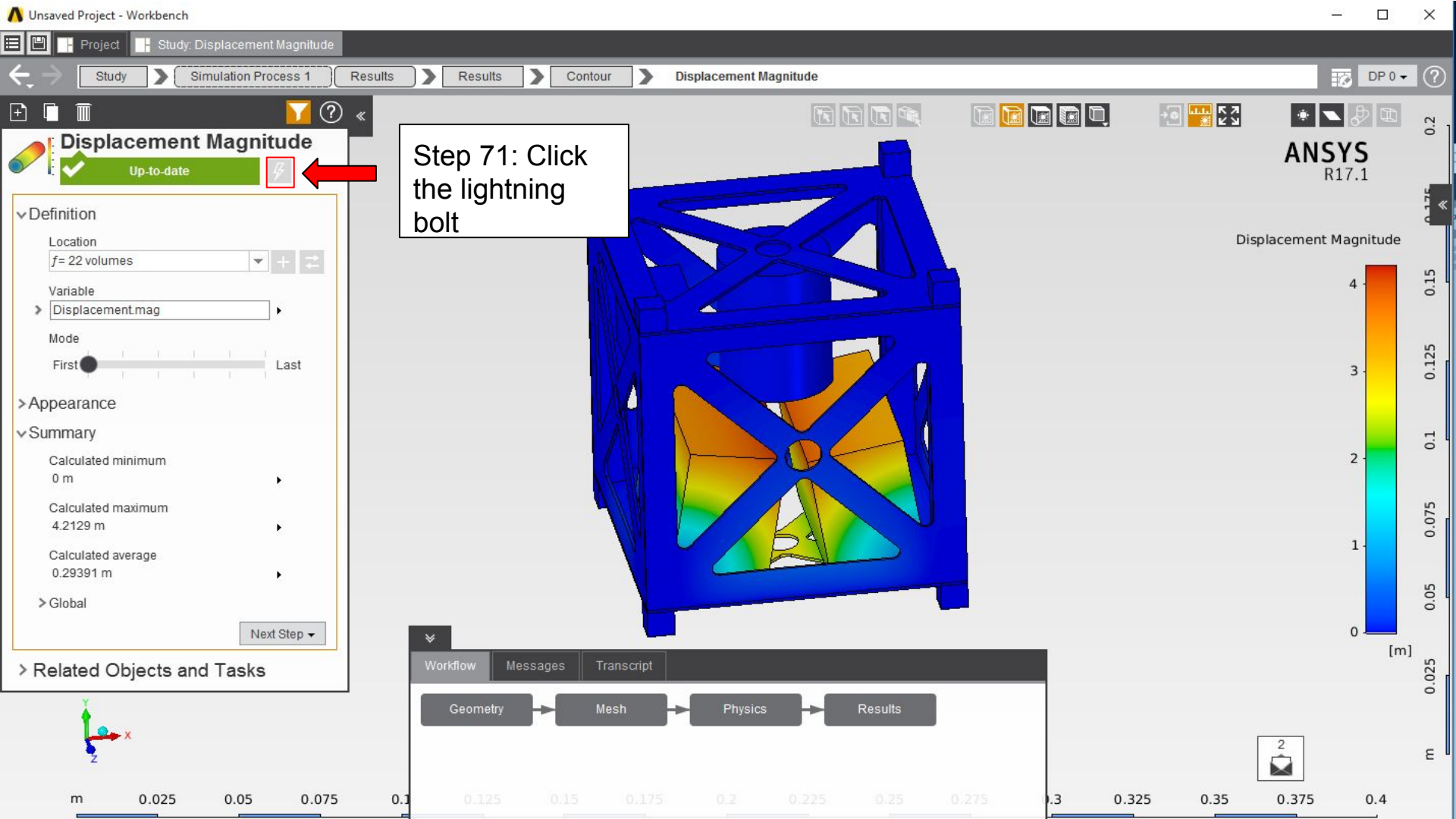

Unsaved Project - Workbench
Project
Study: Aluminum
Study
Simulation Process 1
Physics
Behaviors
Material
Aluminum
DP 0
Aluminum
Up-to-date
ANSYS
R17.1
Density, ρ
2700 kg m^-3
Isotropic Secant Coefficient of Thermal Expansion
Coefficient of thermal expansion, α
2.33E-05 C^-1
Specific heat, c_p
871 J kg^-1 C^-1
Isotropic thermal conductivity, k
237.5 W m^-1 C^-1
Isotropic resistivity
2.6316E-08 ohm m
Isotropic Elasticity
Constant
Derive from
Young's modulus and Poisson's ra
Young's modulus, E
6.89E+10 Pa
Poisson's ratio, v
0.33
Bulk modulus, K
Step 53: Change Density to 2700
Step 54: change young's modulus to 6.89E+10
Step 55: Change Poison's ratio to .33
Step 56: Click on Physics
eometry
Mesh
Physics
0.2
0.15
0.125
0.1
0.075
0.05
0.025
m
0.3
0.325
0.35
0.375
0.4

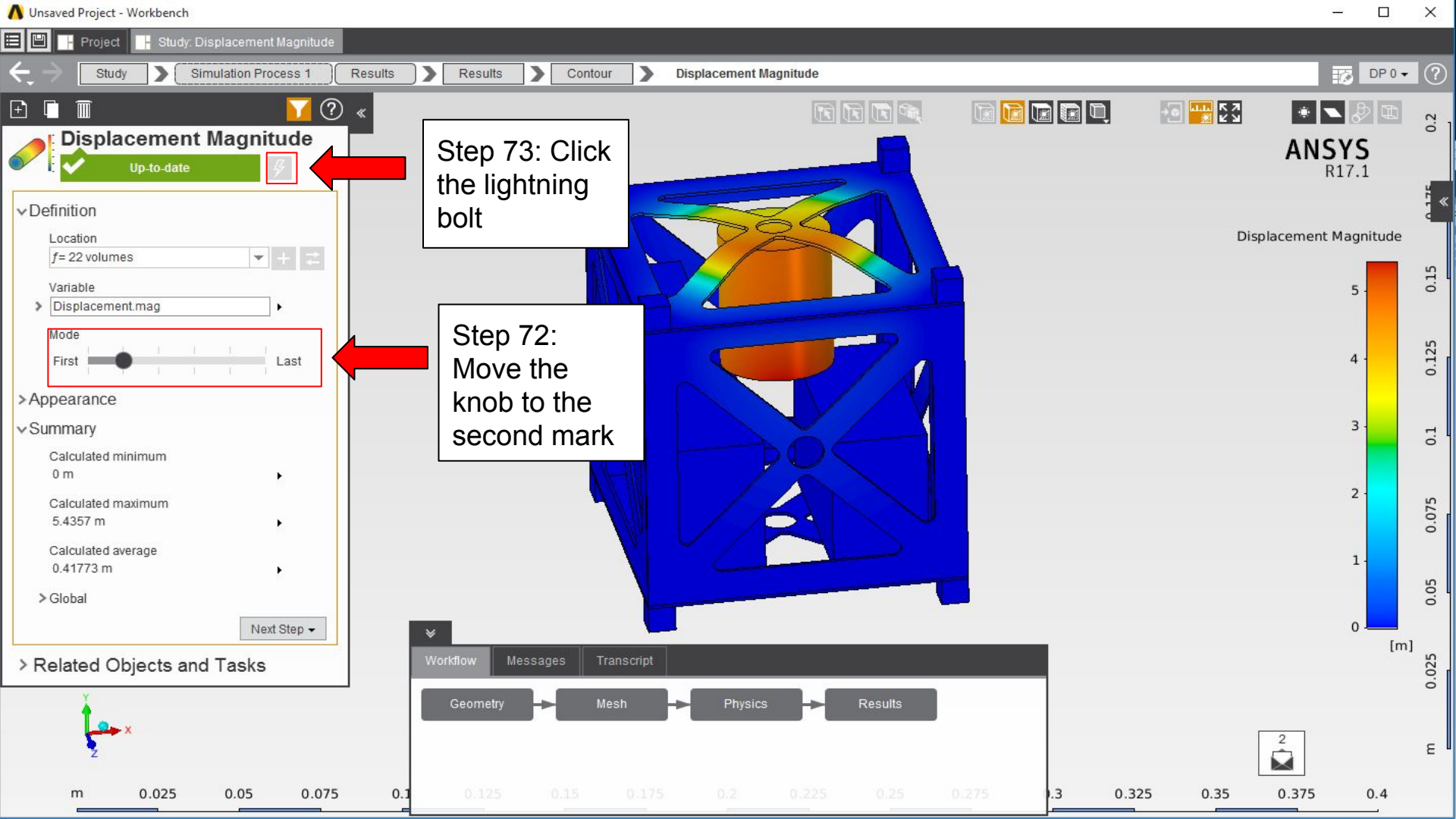

Unsaved Project - Workbench
Project
Study: Physics
Study
Simulation Process 1
Physics
DP 0
Physics
Out-of-date
Settings
Calculation type
Modal
Physics Definition
Physics Regions (1)
Add
Material Assignments (4)
Add
Physics Options (1)
Add
Structural Conditions
Add
Interface Conditions (41)
Displacement
Support
Physics Solution
Solver Options (3)
Add
Monitors (1)
Add
Auxiliary Definitions
Output
Solve Physics
Physics has not been solved.
Related Objects
Step 57: Click on “Add”
Step 58:Click Support
ANSYS
R17.1
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4

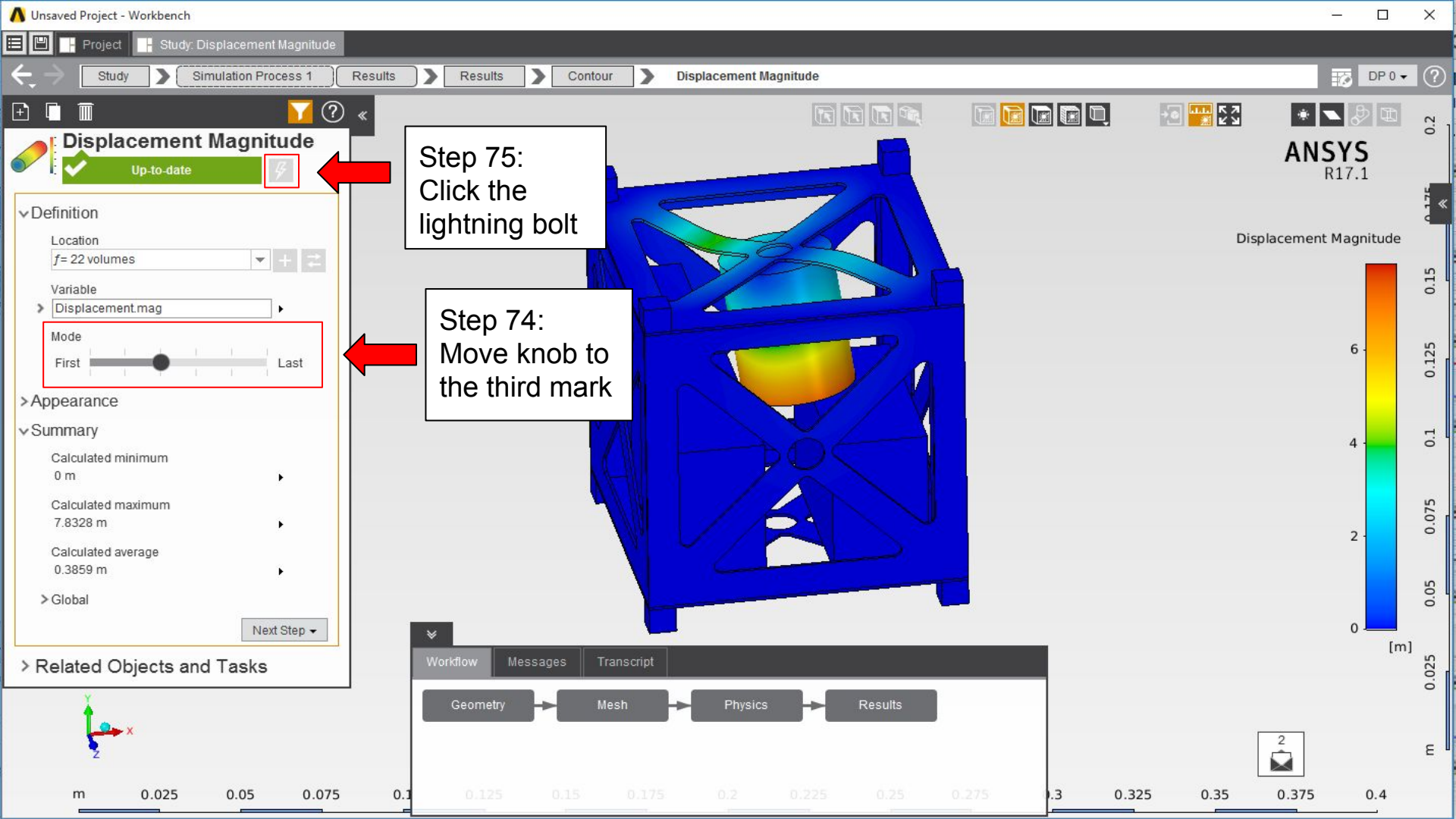

Unsaved Project - Workbench
Project
Study: Support 1
Study
Simulation Process 1
Physics
Structural Conditions
Support
Support 1
DP 0
Support 1
Up-to-date
Location
4 faces
Type
Fixed
Next Step
Related Objects and
Add
Fix
Go to
Physics Modeli
Structural Conditions
Interface Conditions
Selection Sets
Reference Frames
Construction Geometry
Monitors
Solver Options
Displacement
Support
Du te of
Step 59: Choose face select
Step 60: click on bottom four faces of the cube
Step 61: Click the “+”
Step 62: Click “next step
Step 63: Click “Displacement”
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4
0.425

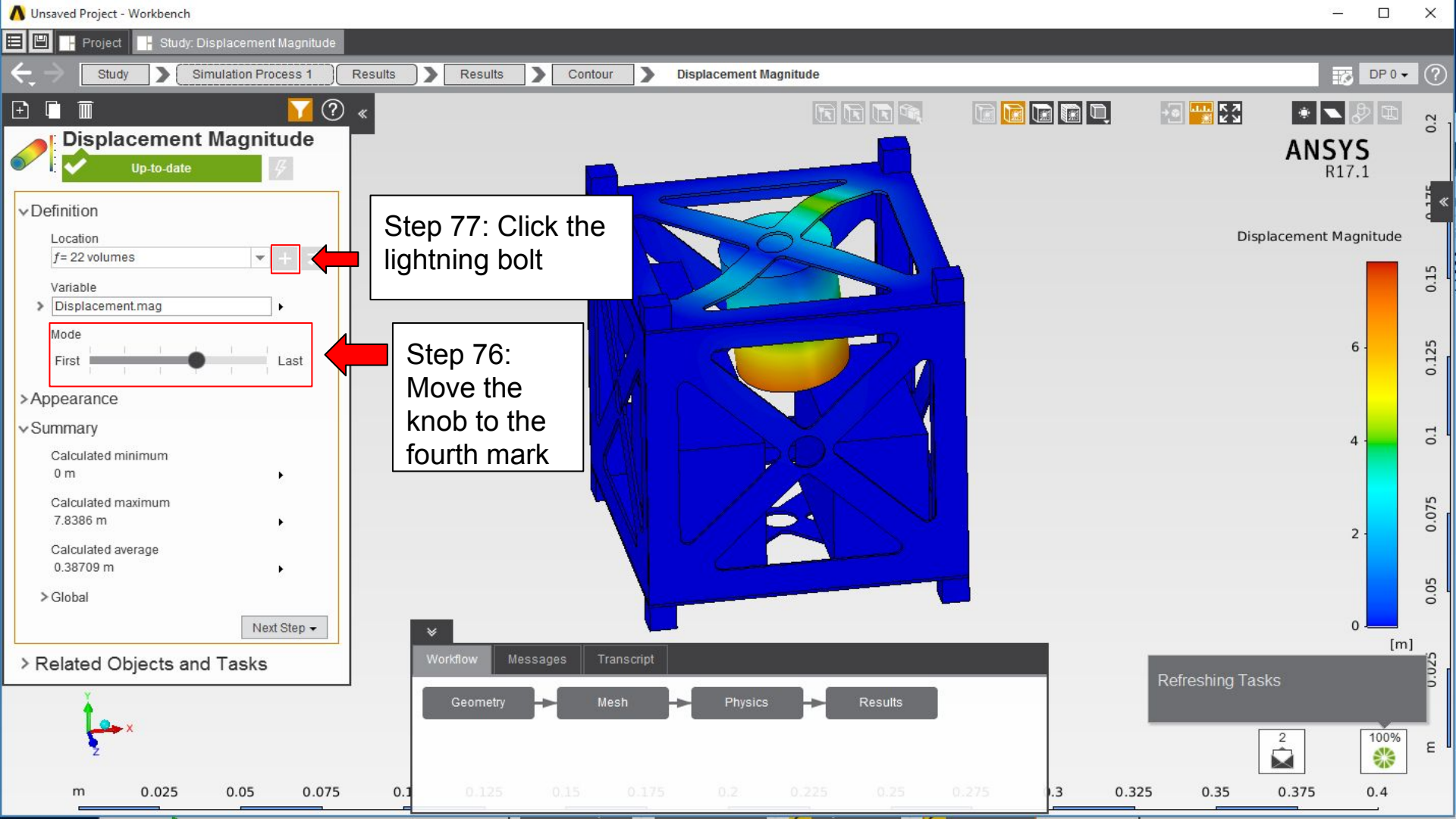

Unsaved Project - Workbench
Project
Study: Displacement 1
Study
Simulation Process 1
Physics
Structural Conditions
Displacement
Displacement 1
DP 0
Displacement 1
Up-to-date
Location
4 faces
Displacement specification
Translation and rotation componer
Apply remotely from originating point
Reference frame
Global Reference Frame
Translation X
0 m
Translation Y
Free
Translation Z
0 m
Rotation X
Free
Rotation Y
Free
Rotation Z
Free
Next Step
Related Objects and Tasks
ANSYS
R17.1
Step 65: Click "+"
Step 64: Click on the top four faces
Step 66: type in "free"
Step 68: Click "Results"
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
Update
Delete
Duplicate
Step 67: Right Click on Results , then Click "Update"
m
0.025
0.05
0.075
0.1
0.325
0.35

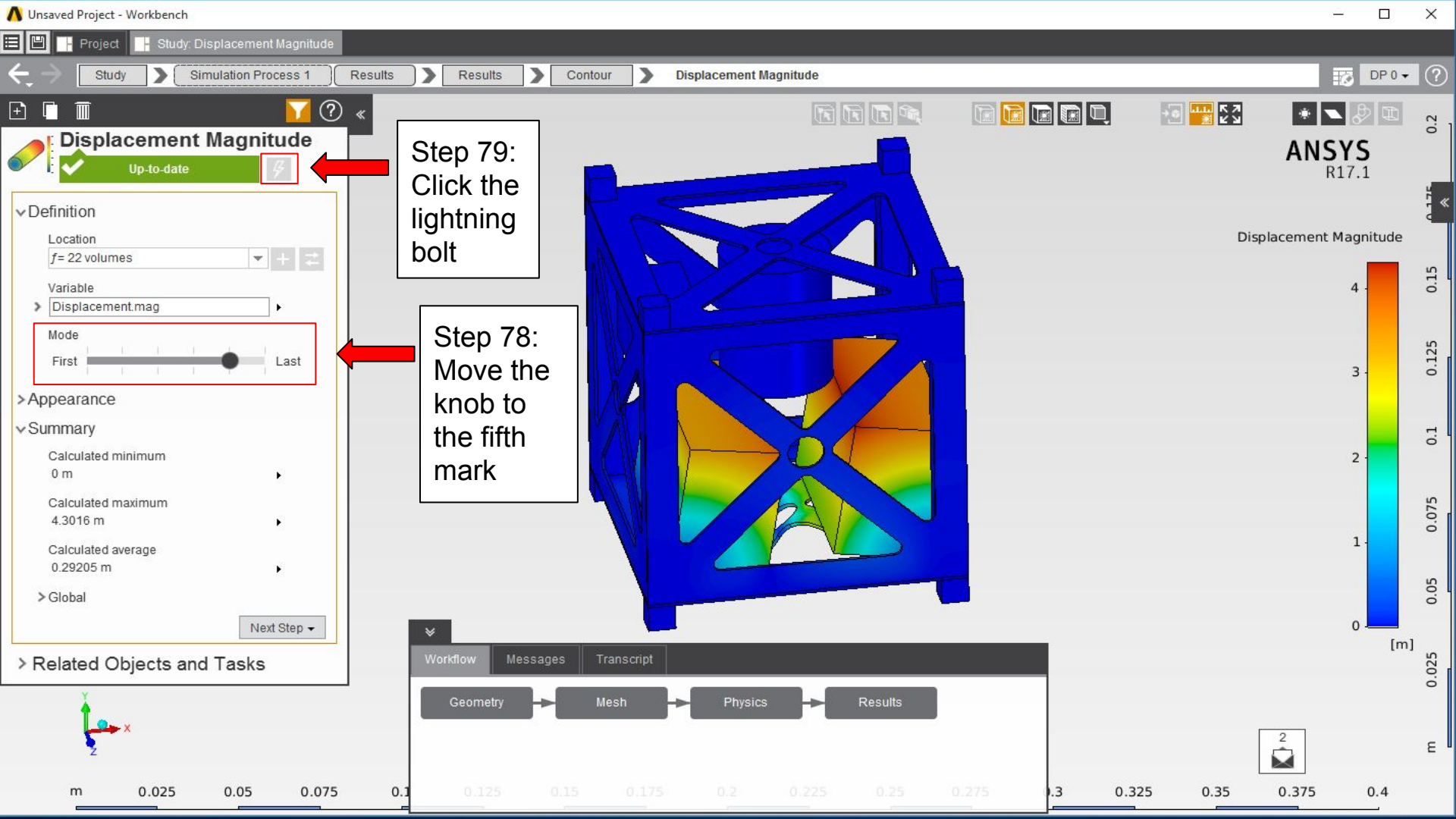

Unsaved Project - Workbench
Project
Study: Results
Study
Simulation Process 1
Results
DP 0
Results
Out-of-date
Settings
Graphics Control
Objects
Results
Add
Displacement Magnitude
Contour
Calculated Value
Isosurface
Vector
Auxiliary Definitions
Selection Sets
Reference Frames
Construction Geometry
Output
Evaluate Results
Results have not been evaluated
Related Objects
Step 69: Click “add”
Step 70: Click “Displacement Magnitude”
ANSYS
R17.1
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4
0.425

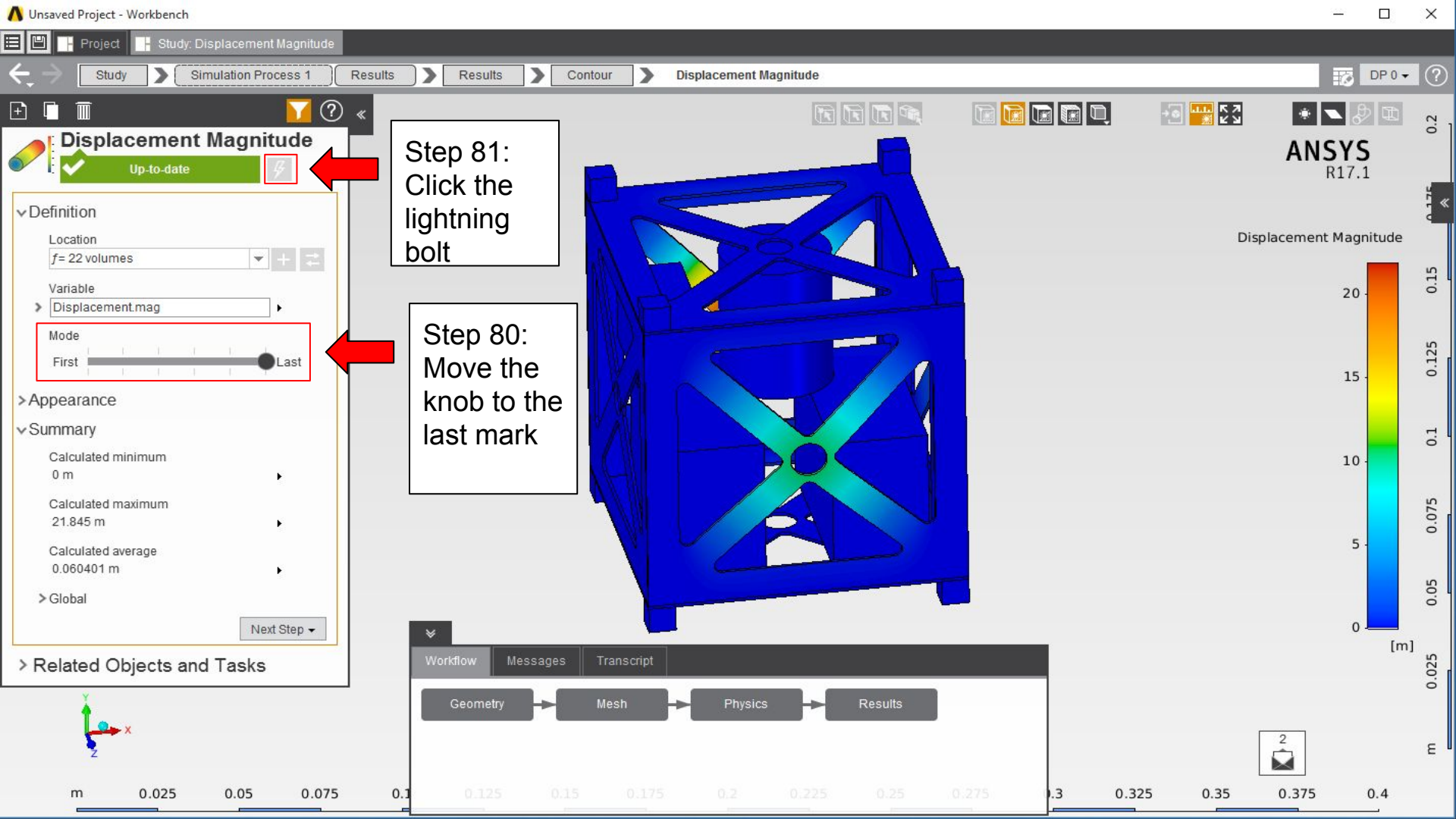

Unsaved Project - Workbench
Project
Study: Displacement Magnitude
Study
Simulation Process 1
Results
Results
Contour
Displacement Magnitude
DP 0
Displacement Magnitude
Up-to-date
Definition
Location
f = 22 volumes
Variable
Displacement.mag
Mode
First
Last
Appearance
Summary
Calculated minimum
0 m
Calculated maximum
4.2129 m
Calculated average
0.29391 m
Global
Next Step
Related Objects and Tasks
Step 71: Click the lightning bolt
ANSYS
R17.1
Displacement Magnitude
4
3
2
1
0
[m]
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4


Unsaved Project - Workbench
Project
Study: Displacement Magnitude
Study
Simulation Process 1
Results
Results
Contour
Displacement Magnitude
DP 0
Displacement Magnitude
Up-to-date
Definition
Location
f= 22 volumes
Variable
Displacement.mag
Mode
First
Last
Appearance
Summary
Calculated minimum
0 m
Calculated maximum
5.4357 m
Calculated average
0.41773 m
Global
Next Step
Related Objects and Tasks
Step 73: Click the lightning bolt
Step 72: Move the knob to the second mark
ANSYS
R17.1
Displacement Magnitude
5
4
3
2
1
0
[m]
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4


Unsaved Project - Workbench
Project
Study: Displacement Magnitude
Study
Simulation Process 1
Results
Results
Contour
Displacement Magnitude
DP 0
Displacement Magnitude
Up-to-date
Definition
Location
f= 22 volumes
Variable
Displacement.mag
Mode
First
Last
Appearance
Summary
Calculated minimum
0 m
Calculated maximum
7.8328 m
Calculated average
0.3859 m
Global
Next Step
Related Objects and Tasks
Step 75: Click the lightning bolt
Step 74: Move knob to the third mark
ANSYS
R17.1
Displacement Magnitude
6
4
2
0
[m]
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.4


Unsaved Project - Workbench
Project
Study: Displacement Magnitude
Study
Simulation Process 1
Results
Results
Contour
Displacement Magnitude
DP 0
Displacement Magnitude
Up-to-date
Definition
Location
f= 22 volumes
Variable
Displacement.mag
Mode
First
Last
Appearance
Summary
Calculated minimum
0 m
Calculated maximum
7.8386 m
Calculated average
0.38709 m
Global
Next Step
Related Objects and Tasks
Step 77: Click the lightning bolt
Step 76: Move the knob to the fourth mark
ANSYS
R17.1
Displacement Magnitude
6
4
2
0
[m]
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
Refreshing Tasks
2
100%
m
0.025
0.05
0.075
0.325
0.35
0.375
0.4


Unsaved Project - Workbench
Project
Study: Displacement Magnitude
Study
Simulation Process 1
Results
Results
Contour
Displacement Magnitude
DP 0
Displacement Magnitude
Up-to-date
Definition
Location
f= 22 volumes
Variable
Displacement.mag
Mode
First
Last
Appearance
Summary
Calculated minimum
0 m
Calculated maximum
4.3016 m
Calculated average
0.29205 m
Global
Next Step
Related Objects and Tasks
Step 79: Click the lightning bolt
Step 78: Move the knob to the fifth mark
ANSYS
R17.1
Displacement Magnitude
4
3
2
1
0
[m]
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4


Unsaved Project - Workbench
Project
Study: Displacement Magnitude
Study
Simulation Process 1
Results
Results
Contour
Displacement Magnitude
DP 0
Displacement Magnitude
Up-to-date
Definition
Location
f= 22 volumes
Variable
Displacement.mag
Mode
First
Last
Appearance
Summary
Calculated minimum
0 m
Calculated maximum
21.845 m
Calculated average
0.060401 m
Global
Next Step
Related Objects and Tasks
Step 81: Click the lightning bolt
Step 80: Move the knob to the last mark
ANSYS
R17.1
Displacement Magnitude
20
15
10
5
0
[m]
Workflow
Messages
Transcript
Geometry
Mesh
Physics
Results
m
0.025
0.05
0.075
0.1
0.125
0.15
0.175
0.2
0.225
0.25
0.275
0.3
0.325
0.35
0.375
0.4