



ESCOLA POLITÉCNICA DA UNIVERSIDADE DE SÃO PAULO

Elementos de Máquinas para Automação

PMR 3307 – A06 / A07

Tutorial para o Ansys

Falha por fadiga mecânica

2020.2



Exercício Aulas 06 e 07

- ▶ Formar 16 equipes de 4 alunos e 2 de 6 alunos
- ▶ A tabela abaixo designa um modelo de chapa (arquivo *.IGES) com concentrador de tensões para cada equipe, assim como o material a ser ensaiado.

Nome	Peças	Material
Equipe 1	Concentrador-1, 1-F e 1-G	Al 6061-T6
Equipe 2	Concentrador-2, 2-F e 2-G	Al 7039
Equipe 3	Concentrador-3, 3-F e 3-G	Al 7075-T6
Equipe 4	Concentrador-4, 4-F e 4-G	Al 2024-T4
Equipe 5	Concentrador-5, 5-F e 5-G	Al 2024T351
Equipe 6	Concentrador-6, 6-F e 6-G	Al 5083H116
Equipe 7	Concentrador-7, 7-F e 7-G	SS 21-6-9 (SS= Aço Inoxidável)
Equipe 8	Concentrador-8, 8-F e 8-G	SS-304 (SS= Aço Inoxidável)
Equipe 9	Concentrador-9, 9-F e 9-G	Steel 1006
Equipe 10	Concentrador-10, 10-F e 10-G	Steel 4340
Equipe 11	Concentrador-11, 11-F e 11-G	Steel S 7
Equipe 12	Concentrador-12, 12-F e 12-G	Steel V250
Equipe 13	Concentrador-13R0, R2, R5 e R10	Ti 6%Al 4%V
Equipe 14	Concentrador-14, 14-F e 14-G	<u>Polycarb</u>
Equipe 15	Concentrador-15, 15-F e 15-G	Nylon
Equipe 16	Concentrador-16, 16-F e 16-G	Iron ARMCO2
Equipe 17	Concentrador-17, 17-F e 17-G	Mag. AZ31B
Equipe 18	Concentrador-18, 18-F e 18-G	<u>Nickel 200</u>



Exercício Aulas 06 e 07

- ▶ Utilizando o software ANSYS (modo *dynamic explicit*) realizar uma simulação para cada modelo até o rompimento. Utilizar carregamento em tração.
- ▶ Identificar as condições para início da trinca para cada modelo
- ▶ Utilizando os dados da simulação calcular o valor do concentrador de tensões, e verificar o valor calculado com os valores encontrados na literatura.



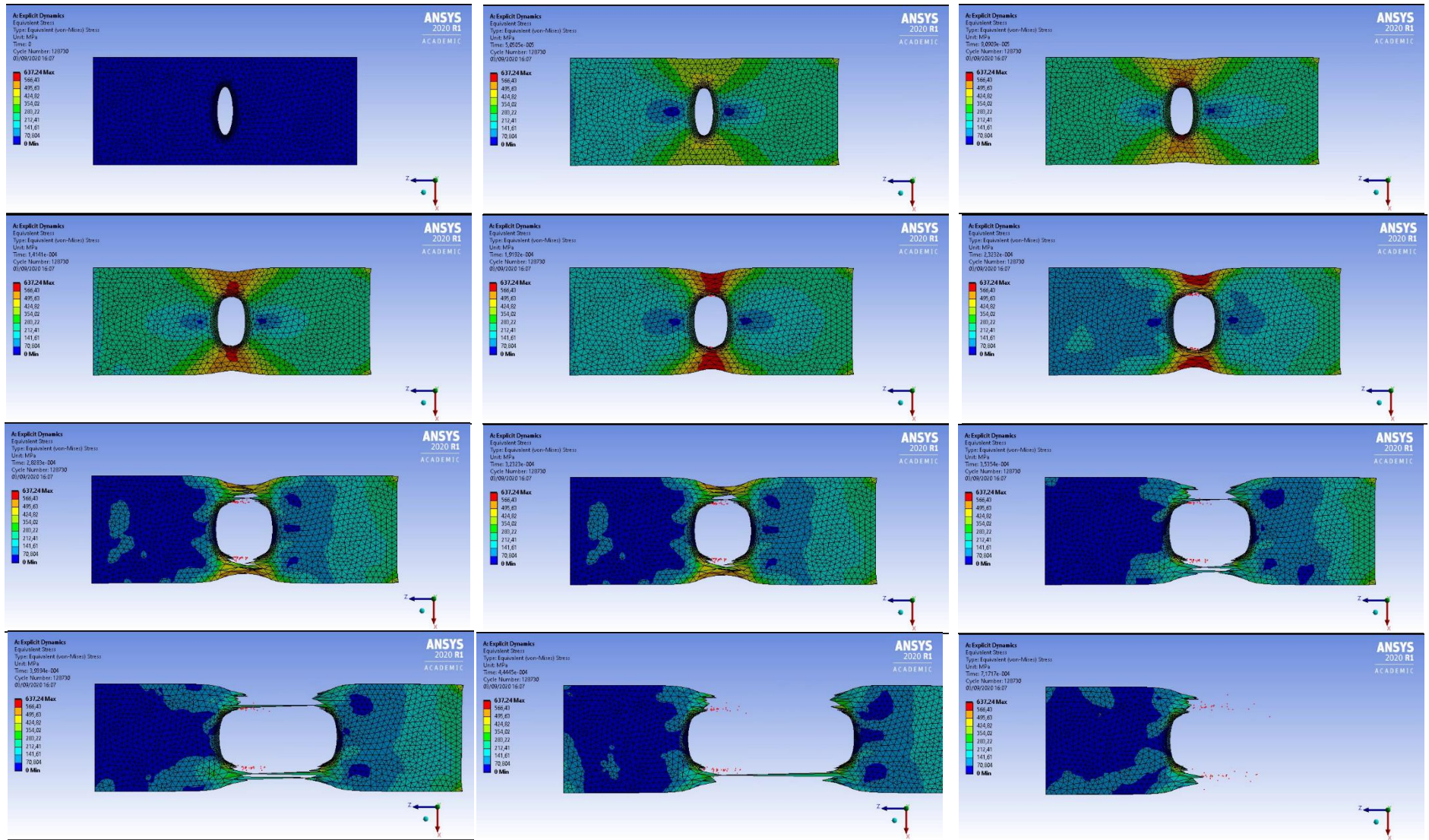
Exercício Aulas 06 e 07

- ▶ Comparar os resultados para chapas finas (-F, esp.=0,3mm), chapas grossas (-G, esp.= 8mm) e normais (esp.= 3mm), exceto para a equipe 13.
- ▶ Equipe 13 ira comparar o efeito do raio de arredondamento, neste caso específico usar carregamento em flexão.

Entrega: até dia 09.10 as 18:00 horas



Não é necessário enviar os vídeos: mas tem que apresentar a sequência conforme exemplo abaixo

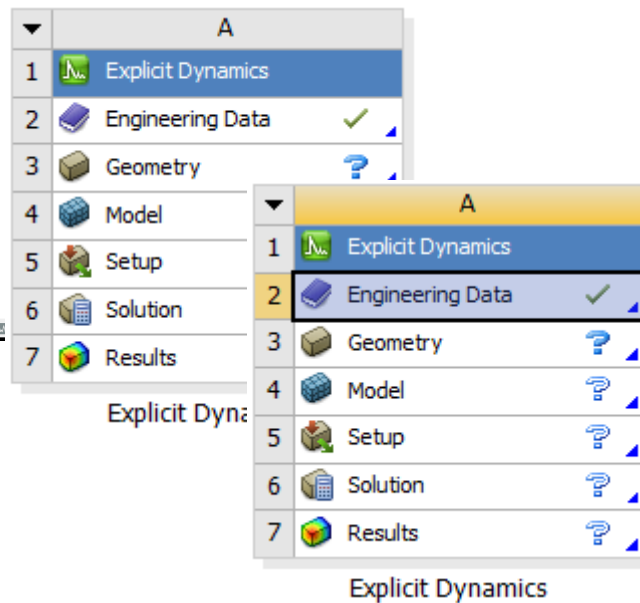
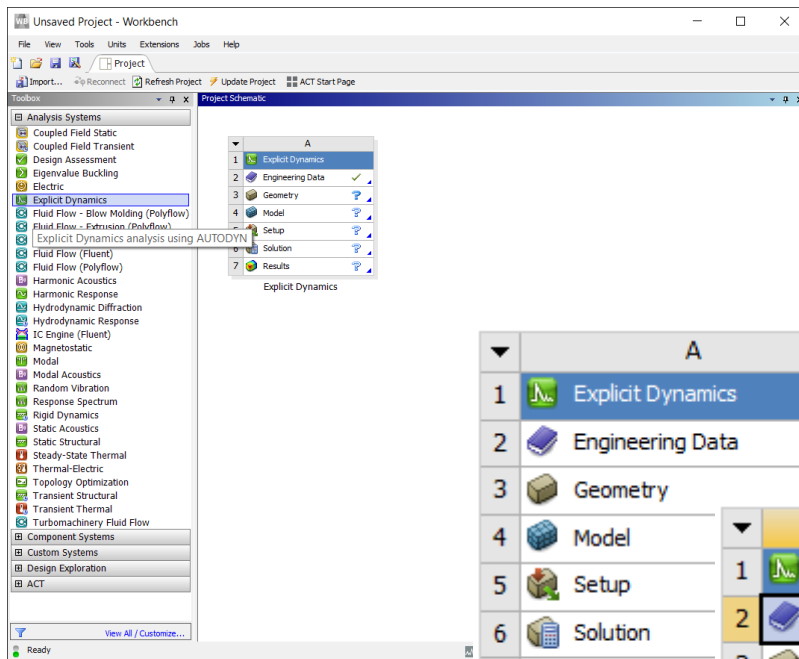




Observação: As equipes deverão ser formadas até 15.09, e seus constituintes informados na aula desta data



Orientações





Orientações

The screenshot shows the ANSYS Workbench interface. On the left, the 'Explicit Dynamics' workflow is selected. The 'Engineering Data Sources' dialog box is open, displaying a table of material data sources. A blue arrow points to the 'Engineering Data Sources' button in the toolbar, and another blue arrow points to the 'Explicit Materials' entry in the dialog box.

	A	B	C	D
1	Data Source		Location	Description
2	★ Favorites			Quick access list and default items
3	Granta Design Sample Materials			Sampling of ANSYS Granta material datasheets. Visit ANSYS GRANTA Materials Data for Simulation to learn about the 6.0 product with...
4	General Materials			Ge...
5	Additive Manufacturing Materials			Ad...
7	Composite Materials	<input type="checkbox"/>		Material samples specific for composite structures.
8	General Non-linear Materials	<input type="checkbox"/>		General use material samples for use in non-linear analyses.
9	Explicit Materials	<input type="checkbox"/>		Material samples for use in an explicit analysis.
10	Hyperelastic Materials	<input type="checkbox"/>		Material stress-strain data samples for curve fitting.
11	Magnetic B-H Curves	<input type="checkbox"/>		B-H Curve samples specific for use in a magnetic analysis.
12	Thermal Materials	<input type="checkbox"/>		Material samples specific for use in a thermal analysis.
13	Fluid Materials	<input type="checkbox"/>		Material samples specific for use in a fluid analysis.
*	Click here to add a new library			



Orientações

	A	B	C	D
1	Data Source		Location	Description
				use with geomechanical models.
7	Composite Materials	<input type="checkbox"/>		Material samples specific for composite structures.
8	General Non-linear Materials	<input type="checkbox"/>		General use material samples for use in non-linear analyses.
9	Explicit Materials	<input type="checkbox"/>		Material samples for use in an explicit analysis.
10	Hyperelastic Materials	<input type="checkbox"/>		Material stress-strain data samples for curve fitting.
11	Magnetic B-H Curves	<input type="checkbox"/>		B-H Curve samples specific for use in a magnetic analysis.
12	Thermal Materials	<input type="checkbox"/>		Material samples specific for use in a thermal analysis.
13	Fluid Materials	<input type="checkbox"/>		Material samples specific for use in a fluid analysis.
*	Click here to add a new library			

	A	B	C	D
1	Data Source		Location	Description
				use with geomechanical models.
7	Composite Materials	<input type="checkbox"/>		Material samples specific for composite structures.
8	General Non-linear Materials	<input type="checkbox"/>		General use material samples for use in non-linear analyses.
9	Explicit Materials	<input type="checkbox"/>		Material samples for use in an explicit analysis.
10	Hyperelastic Materials	<input type="checkbox"/>		Material stress-strain data samples for curve fitting.
11	Magnetic B-H Curves	<input type="checkbox"/>		B-H Curve samples specific for use in a magnetic analysis.
12	Thermal Materials	<input type="checkbox"/>		Material samples specific for use in a thermal analysis.
13	Fluid Materials	<input type="checkbox"/>		Material samples specific for use in a fluid analysis.
*	Click here to add a new library			

	A	B	C	D
1	Contents of Explicit Materials	Add		Source
4	Air (Atmospheric)			Explicit_Materials.xml
5	AL 1100-O			Explicit_Materials.xml

Adicionar



Orientações

	A	B	C	D
1	Data Source		Location	Description
7	Composite Materials	<input type="checkbox"/>		
8	General Non-linear Materials	<input type="checkbox"/>		
9	Explicit Materials	<input type="checkbox"/>		
10	Hyperelastic Materials	<input type="checkbox"/>		
11	Magnetic B-H Curves	<input type="checkbox"/>		
12	Thermal Materials	<input type="checkbox"/>		
13	Fluid Materials	<input type="checkbox"/>		
*	Click here to add a new library			

	A	B	C	D
1	Data Source		Location	Description
7	Composite Materials	<input type="checkbox"/>		Material samples specific for composite structures.
8	General Non-linear Materials	<input type="checkbox"/>		General use material samples for use in non-linear analyses.
9	Explicit Materials	<input type="checkbox"/>		Material samples for use in an explicit analysis.
10	Hyperelastic Materials	<input type="checkbox"/>		Material stress-strain data samples for curve fitting.
11	Magnetic B-H Curves	<input type="checkbox"/>		B-H Curve samples specific for use in a magnetic analysis.
12	Thermal Materials	<input type="checkbox"/>		Material samples specific for use in a thermal analysis.
13	Fluid Materials	<input type="checkbox"/>		Material samples specific for use in a fluid analysis.
*	Click here to add a new library			

	A	B	C	D
1	Contents of Explicit Materials		Add	Source
4	Air (Atmospheric)			Explicit_Materials.xml
5	AL 1100-O			Explicit_Materials.xml

Retornar ao projeto

WB Unsaved Project - Workbench

File Edit View Tools **Project** Extensions Jobs Help

Project A2:Engineering Data x

Filter Engineering Data Engineering Data Sources

Toolbox Project Engineering Data Sources

- Physical Properties
 - Density
- Linear Elastic
 - Isotropic Elasticity
 - Orthotropic Elasticity
 - Viscoelastic
- Hyperelastic Experimental Data
- Hyperelastic
- Plasticity

	A
1	Data Source
7	Composite Materials
8	General Non-linear M
9	Explicit Materials
10	Hyperelastic Material

Adicionar



Orientações

Botão direito do mouse

The screenshot shows a software interface with a tree view on the left. The 'Geometry' item is selected and right-clicked, opening a context menu. A red arrow points to the 'Geometry' item in the tree, and a red dashed box highlights the text 'Botão direito do mouse'. A blue arrow points to the 'Import Geometry' option in the menu, which has a sub-menu. The sub-menu is open, showing a 'Browse...' option circled in red, with a blue arrow pointing to it. Other options in the sub-menu include 'Flambagem-1.IGS', 'Concentrador-1E.IGS', 'Concentrador-3.IGS', and 'Concentrador-1-C.IGS'.

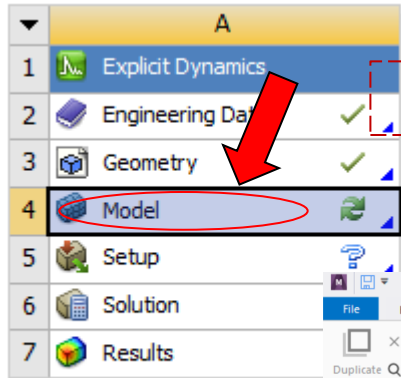
The screenshot shows a software interface with a tree view. The 'Explicit Dynamics' item is selected. Two blue arrows point to the 'Explicit Dynamics' item and the 'Geometry' item in the tree. The tree view shows the following items:

Item	Status
1 Explicit Dynamics	
2 Engineering Data	✓
3 Geometry	✓
4 Model	↻
5 Setup	?
6 Solution	?
7 Results	?

Explicit Dynamics

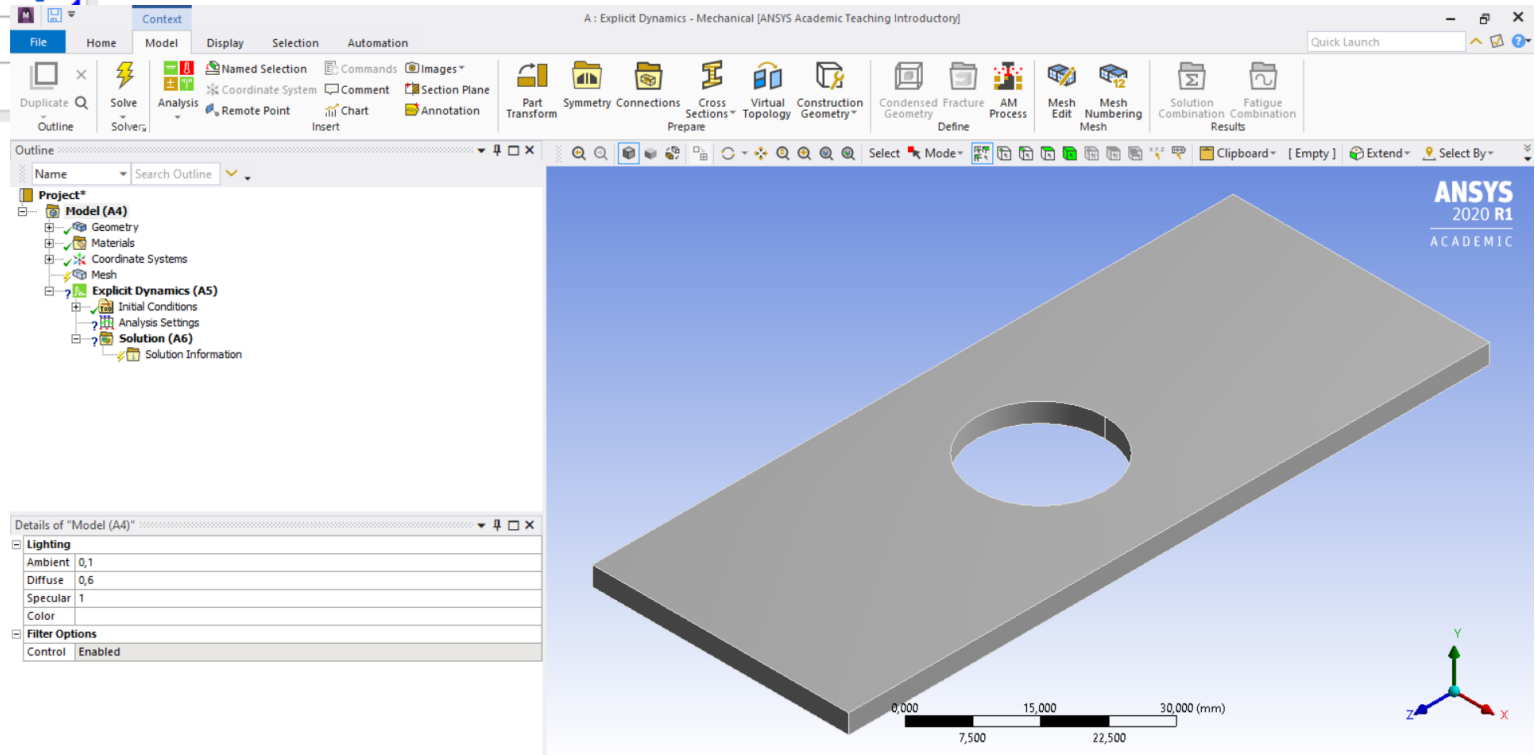


Orientações



Dois clicks botão esquerdo do mouse

Explicit Dynamics





Orientações

The screenshot displays the ANSYS 2020 R1 Academic software interface. The main window shows a 3D model of a mechanical part, a grey plate with a semi-circular hole. The software's ribbon menu is visible at the top, with tabs for File, Home, Context, Model, Display, Selection, and Automation. The Context tab is active, showing various tool icons. The Outline window on the left shows the project hierarchy, with the 'Explicit Dynamics (A5)' folder highlighted by a red circle and a red arrow. The Details of 'Model (A4)' window is open at the bottom left, showing lighting and filter options. A larger, detailed view of the project tree is shown in the bottom right corner, listing the following components:

- Project*
- Model (A4)
 - Geometry
 - Materials
 - Coordinate Systems
 - Mesh
- Explicit Dynamics (A5)
 - Initial Conditions
 - Analysis Settings
- Solution (A6)
 - Solution Information



Orientações

The image illustrates the steps to assign a material to a part in a CAD software. It consists of three screenshots and a materials list:

- Top-left screenshot:** Shows the project tree for 'Project*' with 'Model (A4)' expanded. The 'Geometry' and 'Materials' folders are circled in red, with a red arrow pointing to them.
- Top-right screenshot:** Shows 'Model (A4)' expanded, with 'Concentrador-1-FreeParts' circled in red and a red arrow pointing to it.
- Bottom screenshot:** Shows the 'Details of "Concentrador-1-FreeParts"' dialog box. The 'Material' section is expanded, and the 'Assignment' is set to 'Structural Steel', which is circled in red. A red arrow points to this field.
- Right panel:** Shows the 'Engineering Data Materials' list. 'AL 1100-O' is circled in red, with a red arrow pointing to it.



Orientações

The image illustrates the steps to access the Sizing settings in a CAD application:

- Open the **Project*** tree and expand **Explicit Dynamics (A5)**. The **Mesh** icon is highlighted with a red circle and a red arrow.
- Right-click on **Mesh** and select **Insert** from the context menu. In the **Insert** menu, the **Sizing** option is highlighted with a red circle and a red arrow.
- The **Details of "Sizing" - Sizing** panel is displayed. The **Element Size** property is highlighted with a red circle and a red arrow.

Property	Value
Scoping Method	Geometry Selection
Geometry	No Selection
Definition	
Suppressed	No
Type	Element Size
Element Size	Default (3,367 mm)
Advanced	
Defeature Size	Default
Behavior	Soft



Orientações

Details of "Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	No Selection
Definition	
Suppressed	No
Type	Element Size
<input type="checkbox"/> Element Size	Default (3,367 mm)
Advanced	
<input type="checkbox"/> Defeature Size	Default
Behavior	Soft



A : Explicit Dynamics - Mechanical [ANSYS Academic Teaching Introductory]

File Home Context Display Selection Automation

Duplicate Outline Solve Solver Insert Mesh Update Generate Mesh Surface Mesh Source/Target Preview Method Sizing Face Meshing Mapped Meshing Mesh Copy Controls Refinement Gasket Contact Sizing Inflation Mesh Connection Group Contact Match Group Node Merge Group Mesh Edit Mesh Numbering MeshEdit Manual Mesh Connection Match Merge Node Move Metric Graph Edges Max Min Metrics Display

Outline

Name Search Outline

Project*

- Model (A4)
 - Geometry
 - Concentrador-1-FreeParts
 - Materials
 - Coordinate Systems
 - Mesh
 - Sizing
- Explicit Dynamics (A5)
 - Initial Conditions
 - Analysis Settings
- Solution (A6)
 - Solution Information

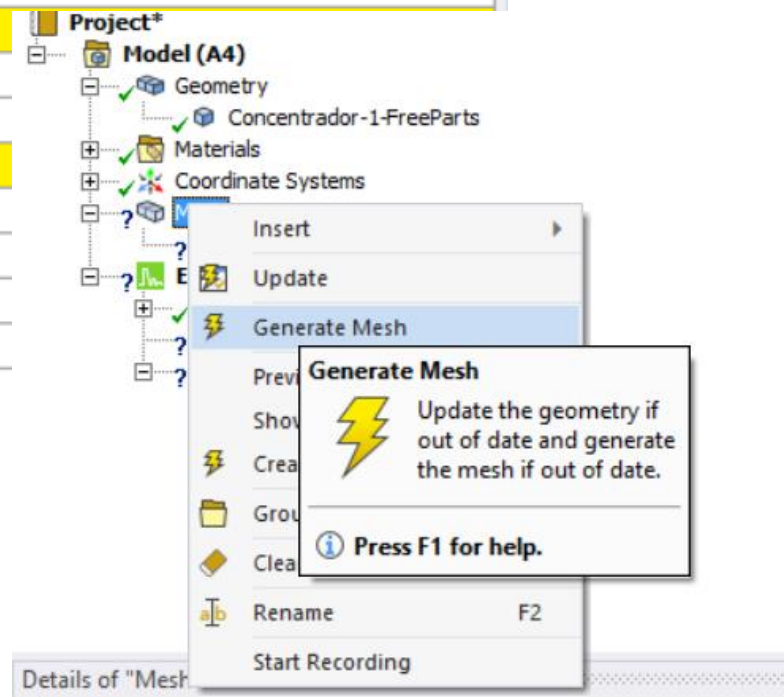
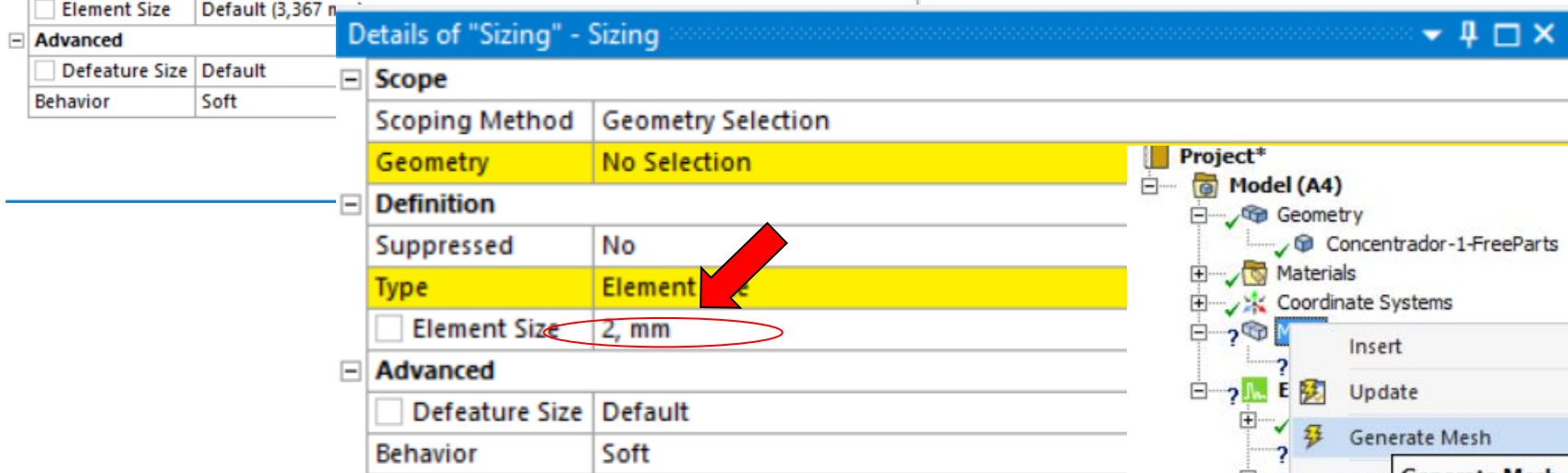
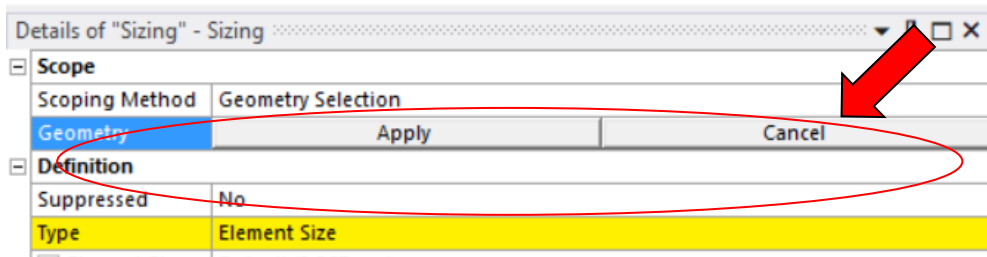
Details of "Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	Apply Cancel
Definition	
Suppressed	No
Type	Element Size
<input type="checkbox"/> Element Size	Default (3,367 mm)
Advanced	
<input type="checkbox"/> Defeature Size	Default
Behavior	Soft

ANSYS 2020 R1 ACADEMIC

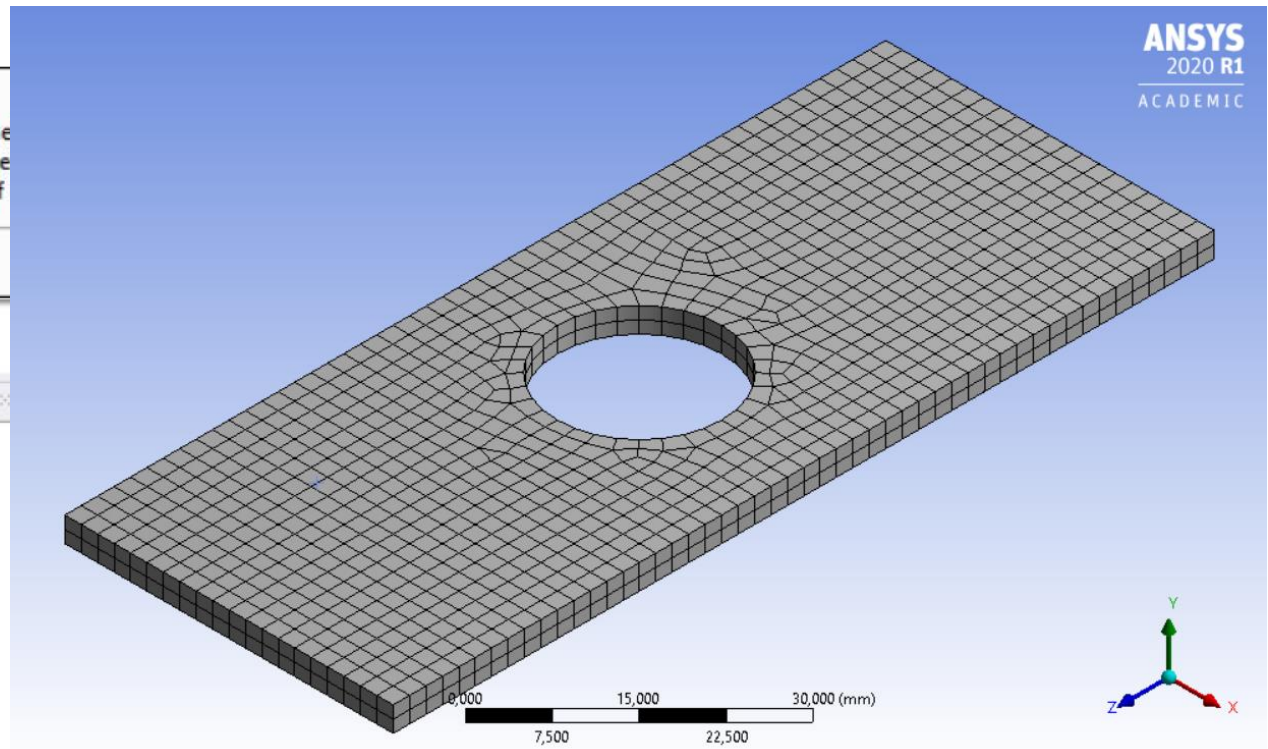
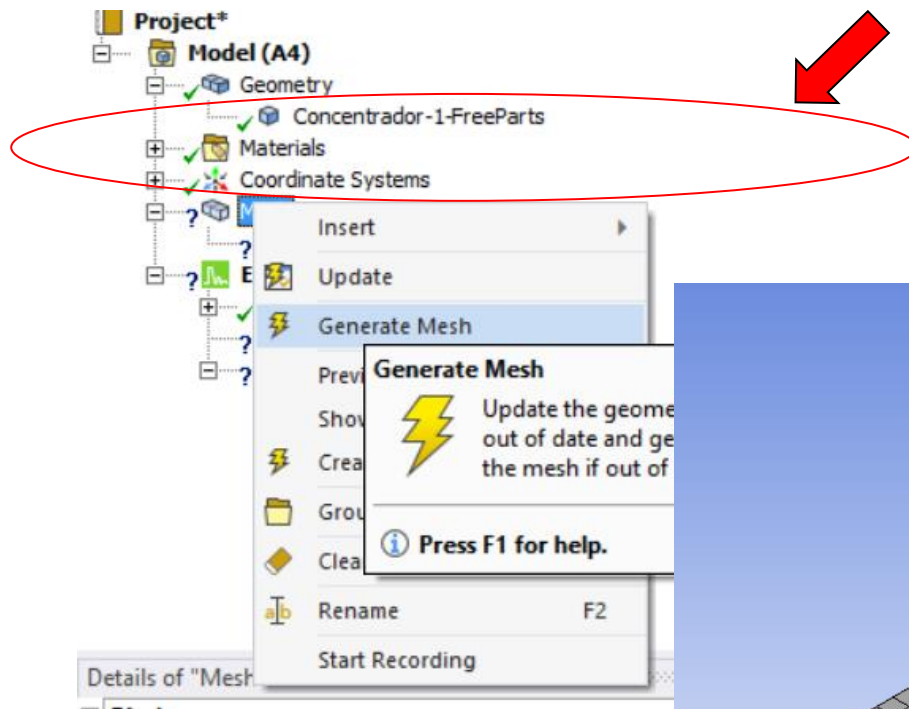


Orientações





Orientações





Orientações

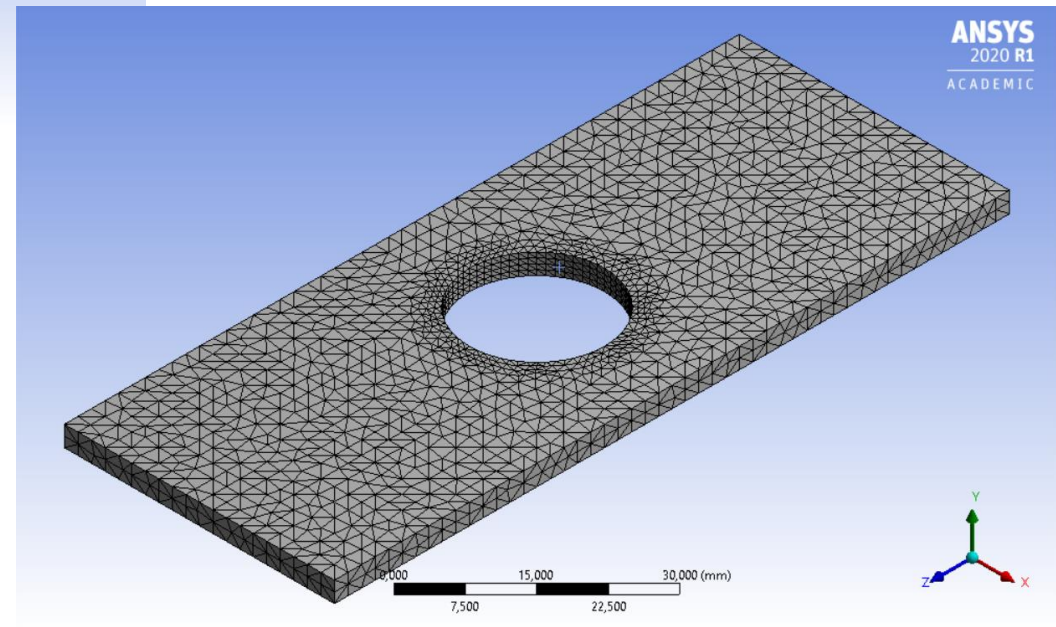
The screenshot shows the ANSYS 2020 R1 software interface. The 'Project*' tree on the left has 'Coordinate Systems' and 'Mesh' circled in red. The 'Mesh' context menu is open, with 'Refinement' circled in red. A tooltip for 'Refinement' is displayed, stating: 'Specify the maximum number of times you want an initial mesh to be refined. You can specify refinement controls for faces, edges, and vertices.' The 'Details of "Mesh"' panel is visible on the left, and the 'Details of "Refinement" - Refinement' panel is open at the bottom, with the 'Geometry Selection' button circled in red.

The screenshot shows a 3D mesh model of a rectangular plate with a circular hole. The mesh is composed of gray quadrilateral elements. A red circle highlights the hole, and a red arrow points to it. The ANSYS 2020 R1 ACADEMIC logo is in the top right corner. The bottom of the image shows a scale bar from 0,000 to 30,000 (mm) and a coordinate system (X, Y, Z).



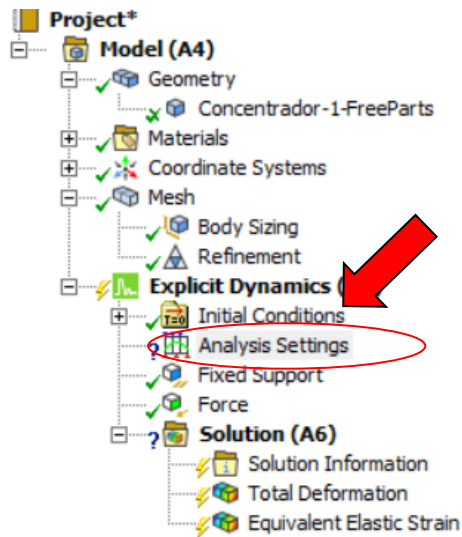
Orientações

The screenshot shows the ANSYS 2020 R1 ACADEMIC interface. The main window displays a 3D model of a rectangular plate with a circular hole, meshed with a fine grid. The 'Project*' tree on the left shows the model structure, including Geometry, Materials, Coordinate Systems, Mesh, and Solution. A red arrow points to the 'Details of Refinement - Refinement' dialog box, which is open and shows the 'Scope' section with 'Geometry Selection' selected. Another red arrow points to the 'Update' button in the 'Project*' tree, which is highlighted with a red circle. A tooltip for the 'Update' button is visible, stating: 'Update the geometry if out of date and generate the mesh if out of date. Press F1 for help.'





Orientações



Details of "Analysis Settings"

Analysis Settings Preference	
Type	Program Controlled
Step Controls	
Number Of Steps	1
Current Step Number	1
Load Step Type	Explicit Time Integration
End Time	0,001
Resume From Cycle	0
Maximum Number of Cycles	1e+07
Maximum Energy Error	0,1
Reference Energy Cycle	0
Initial Time Step	Program Controlled
Minimum Time Step	Program Controlled



Orientações

The image shows a screenshot of the ANSYS software interface. On the left, the 'Explicit Dynamics' menu is open, with 'Fixed Support' highlighted. A red circle is drawn around the 'Fixed Support' option. A red arrow points to the 'Fixed Support' option. Another red arrow points to the 'Explicit Dynamics' menu. A third red arrow points to the 'Fixed Support' option in the 'Insert' menu. A tooltip for 'Fixed Support' is visible, stating: 'Fixed Support: Insert a Fixed Support boundary condition to prevent a selected geometric or mesh entity from moving or deforming. Press F1 for help.'

Properties of "Explicit Dynamics (A5)"	
Definition	
Physics Type	Structural
Analysis Type	Explicit Dynamics
Solver Target	AUTODYN
Options	
Environment Temperature	22, °C
Generate Input Only	No



Orientações

Explicit Dynamics

- Insert
- Solve
- Duplicate
- Clear Generated Data
- Delete
- Rename F2
- Group All Similar Children
- Open Solver Files Directory

Force

Insert a Force load that distributes a force vector across one or more topologies.

Press F1 for help.

Details of "Force"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face

Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	0, N (step applied)
Y Component	-10000
Z Component	0, N (step applied)
Suppressed	No

A: Explicit Dynamics

Force

Time: 1, s
11/09/2020 07:30

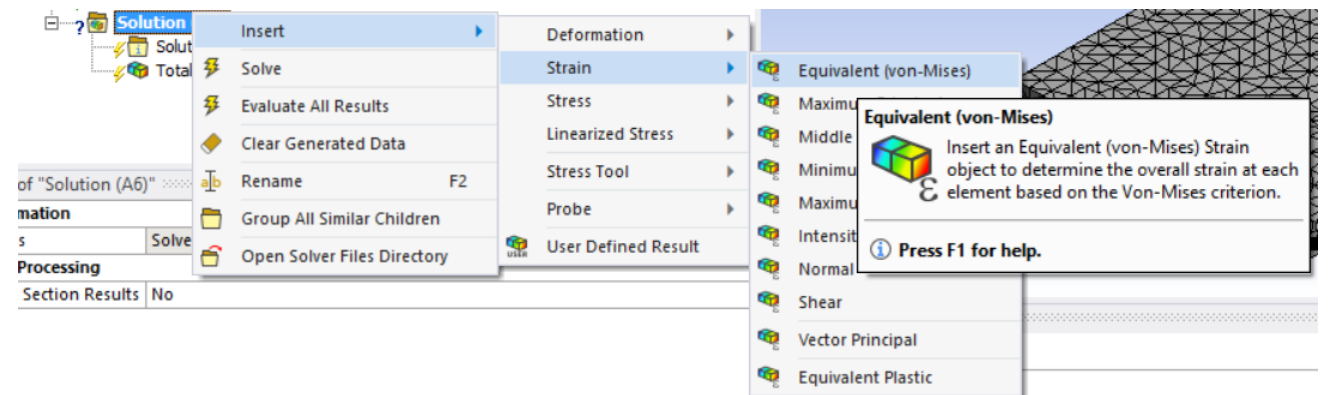
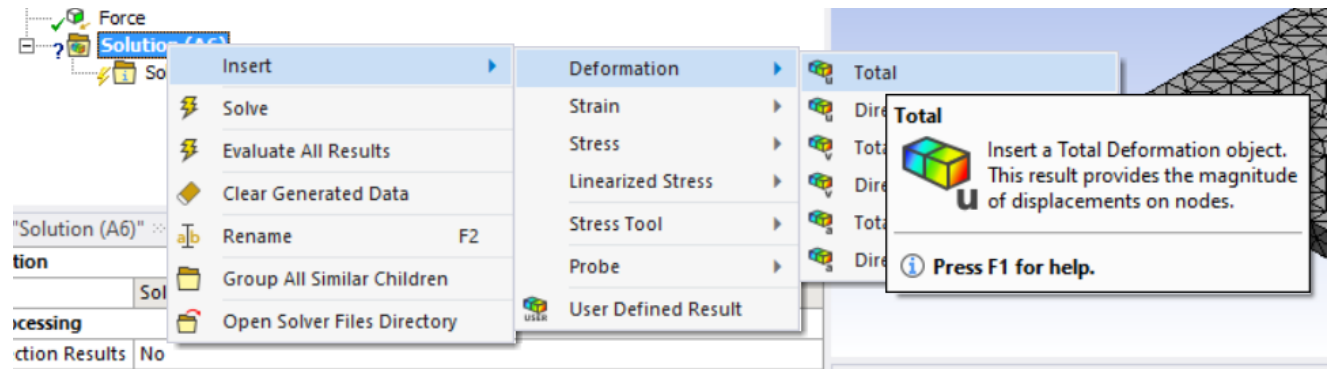
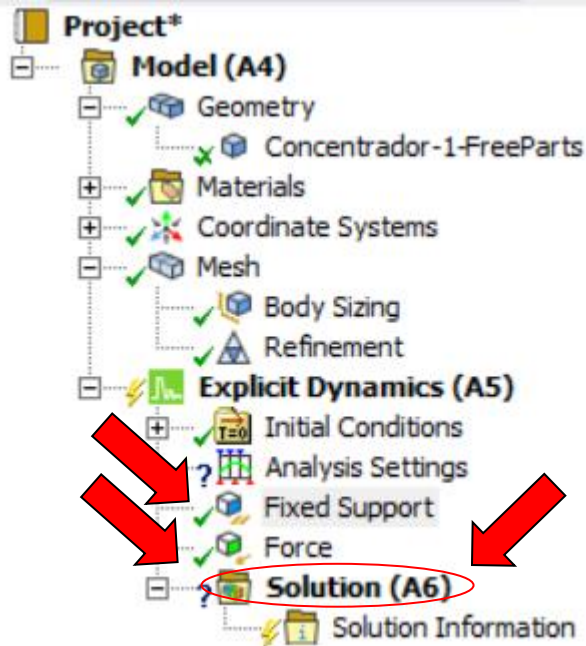
Force: 0, N
Components: 0;0;0, N

Graph

Steps	Time [s]	X [N]	Y [N]	Z [N]
1	1	0	0	0



Orientações





Orientações

