



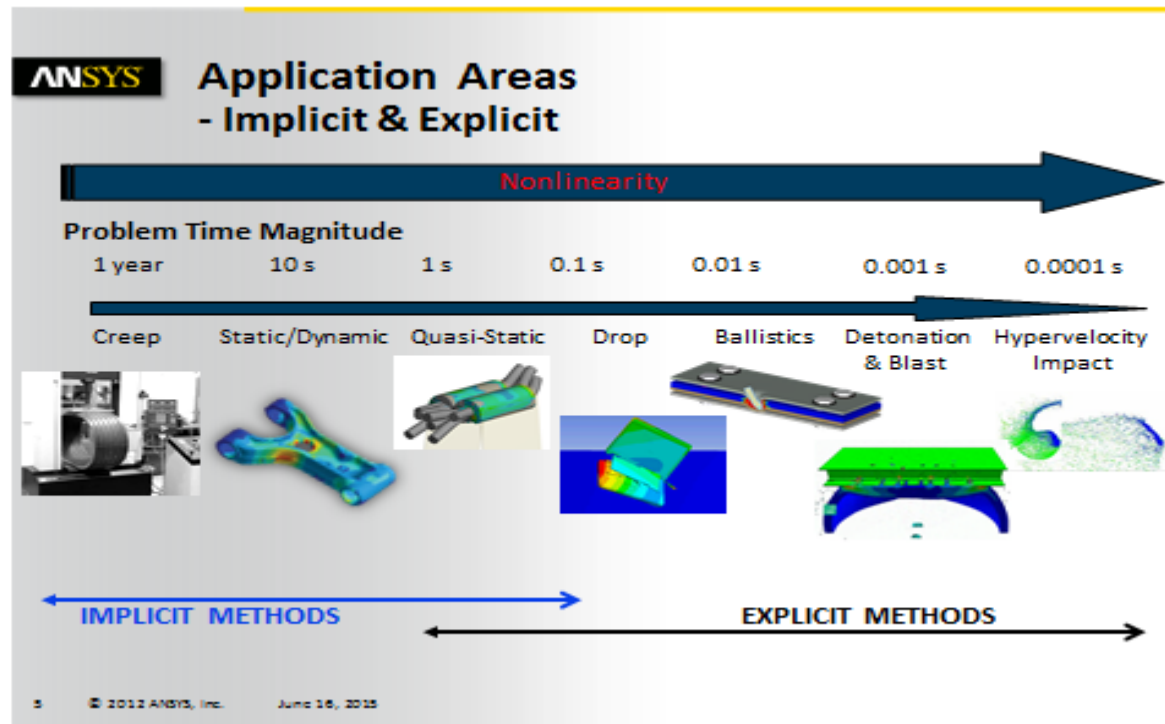
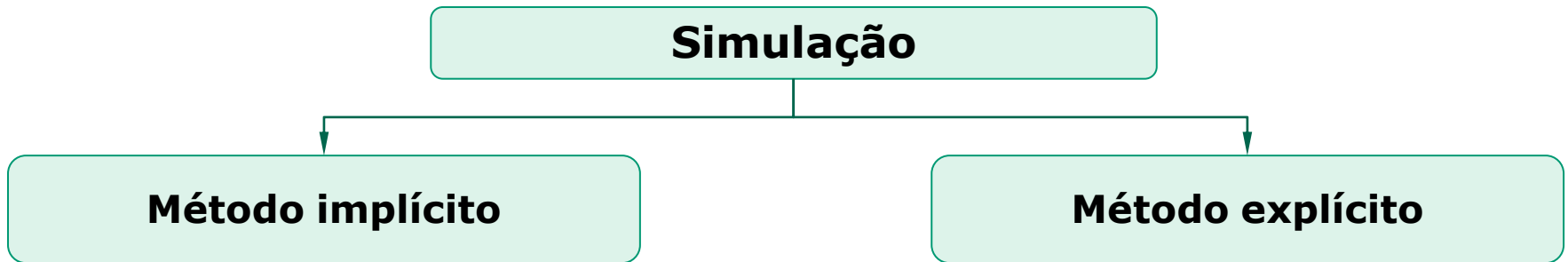
ESCOLA POLITÉCNICA DA UNIVERSIDADE DE SÃO PAULO

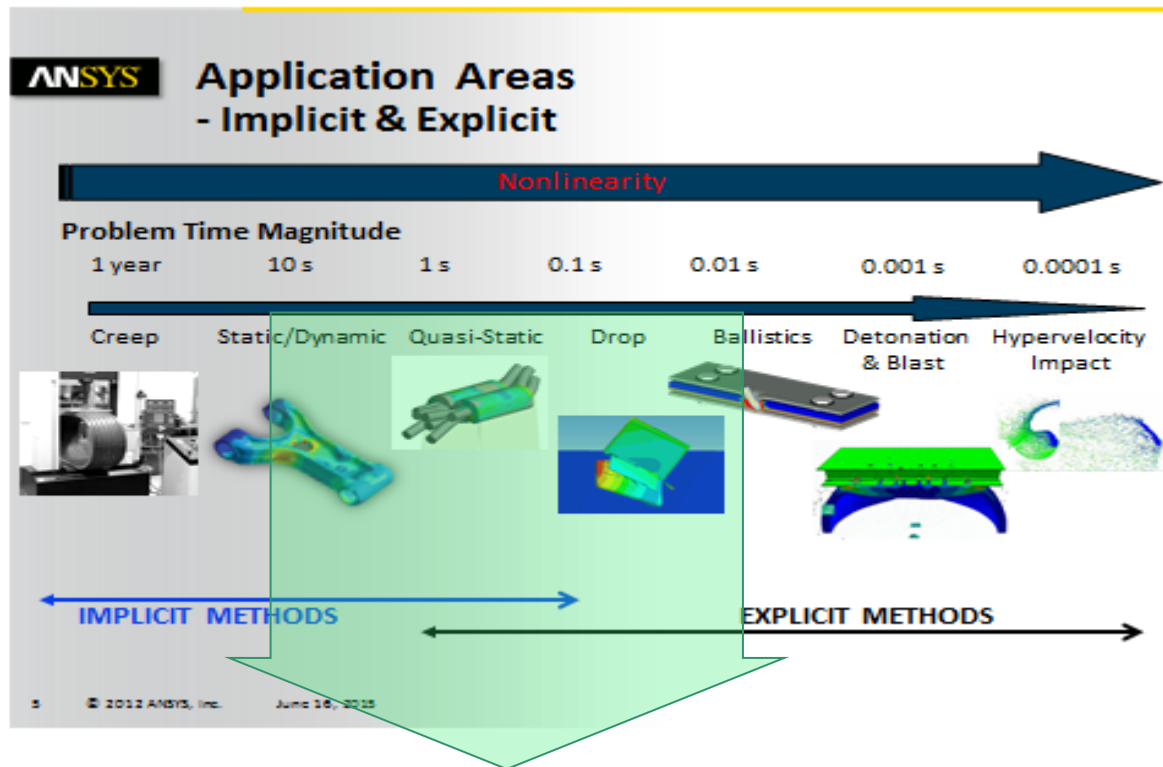
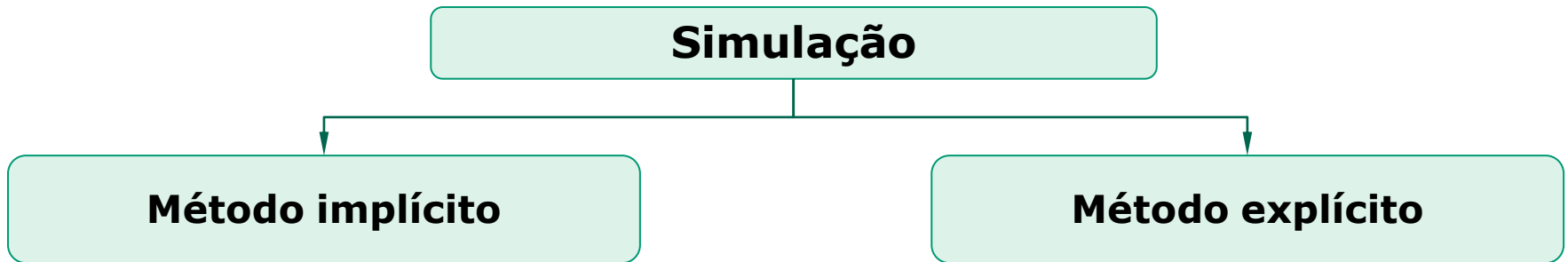
Elementos de Máquinas para Automação

PMR 3307 – Exercício 1

Fator de concentração de tensão

2023.2



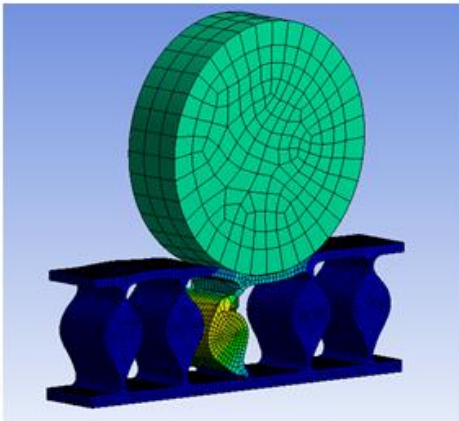


Estamos aqui!



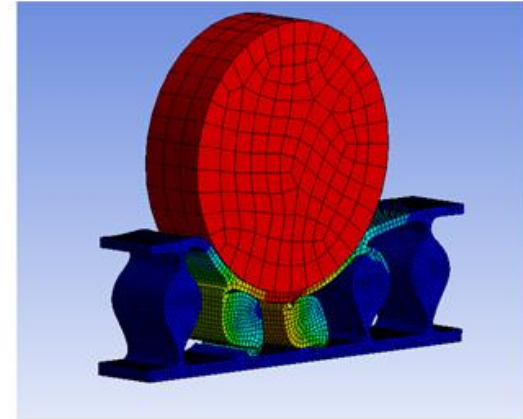
Simulação

Método implícito



$$m\ddot{x} + c\dot{x} + kx = F(t)$$

Método explícito



$$\frac{\rho_0 V_0}{V} = \frac{m}{V} \quad \text{conservation of momentum}$$

$$\rho\ddot{x} = b_x + \frac{\partial\sigma_{xx}}{\partial x} + \frac{\partial\sigma_{xy}}{\partial y} + \frac{\partial\sigma_{xz}}{\partial z}$$

$$\rho\ddot{y} = b_y + \frac{\partial\sigma_{yx}}{\partial x} + \frac{\partial\sigma_{yy}}{\partial y} + \frac{\partial\sigma_{yz}}{\partial z} \quad \text{Conservation of energy}$$

$$\rho\ddot{z} = b_z + \frac{\partial\sigma_{zx}}{\partial x} + \frac{\partial\sigma_{zy}}{\partial y} + \frac{\partial\sigma_{zz}}{\partial z}$$

$$\dot{e} = \frac{1}{\rho} \left(\sigma_{xx}\dot{\epsilon}_{xx} + \sigma_{yy}\dot{\epsilon}_{yy} + \sigma_{zz}\dot{\epsilon}_{zz} + 2\sigma_{xy}\dot{\epsilon}_{xy} + 2\sigma_{yz}\dot{\epsilon}_{yz} + 2\sigma_{zx}\dot{\epsilon}_{zx} \right)$$



Método explícito

O sistema *Explicit Dynamics* é projetado para permitir simulações em mecânica estrutural não linear. É recomendado quando envolvendo uma ou mais das seguintes situações:

- Impacto de baixa [1 m/s] a muito alta velocidade [5000 m/s]
- Propagação da onda de estresse
- Resposta dinâmica de alta frequência
- Grandes deformações e não linearidades geométricas
- Condições de contato complexas
- Comportamento de material complexo, incluindo danos materiais e falhas
- Resposta estrutural não linear, incluindo flambagem.
- Falha de ligações / soldas / fixadores
- Propagação de ondas de choque através de sólidos e líquidos
Corpos rígidos e flexíveis



Método explícito

O método *Explicit Dynamics* é mais adequado para eventos que ocorrem em curtos períodos de tempo, alguns milissegundos ou menos.

Eventos que duram mais de 1 segundo podem ser modelados, no entanto, tempos de processamento serão longos.

Em uma solução de Dinâmica Explícita, começamos com um domínio discretizado (malha) onde são atribuídas: as propriedades do material, as cargas, as restrições e as demais condições iniciais.

Este estado inicial, quando integrado no tempo, irá produzir movimento nos pontos dos nós da malha.



Método explícito

- O movimento dos pontos dos nós produz deformação nos elementos da malha
- A deformação resulta em uma mudança no volume (portanto, densidade) do material em cada elemento
- A taxa de deformação do elemento é usada para derivar as taxas de deformação do material usando várias formulações de elemento
- As leis constitutivas tomam as taxas de deformação do material e derivam as tensões do material resultantes
- As tensões do material são transformadas de volta em forças nodais usando várias formulações de elementos

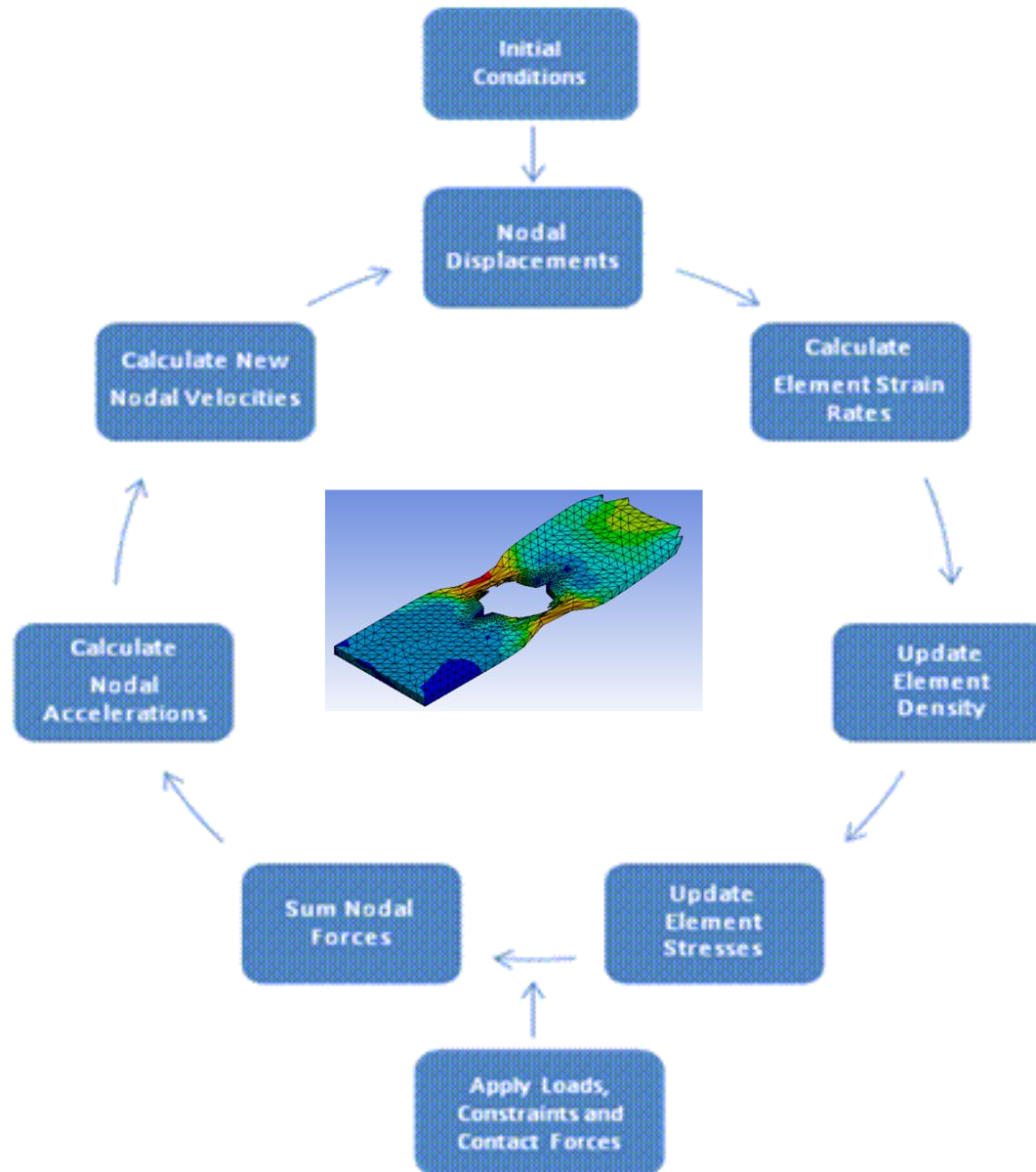


Método explícito

- Forças nodais externas são calculadas a partir de condições de contorno, cargas e contato (interação corporal)
- As forças nodais são divididas pela massa nodal para produzir acelerações nodais
- As acelerações são integradas explicitamente a tempo de produzir novas velocidades nodais
- As velocidades nodais são integradas explicitamente a tempo de produzir novas posições nodais
- O processo de solução (Ciclo) é repetido até que um tempo definido pelo usuário seja alcançado



Método explícito





Exercício 2

- ▶ Utilizando o software ANSYS no modo *Dynamic Explicit*, realizar a simulação de uma placa com concentrador de tensões submetida a um carregamento trativo, até o rompimento.
- ▶ Identificar no modelo as condições para início da trinca.
- ▶ Determinar o fator de concentração de tensões.
- ▶ Comparar o resultados com os valores encontrados na literatura.

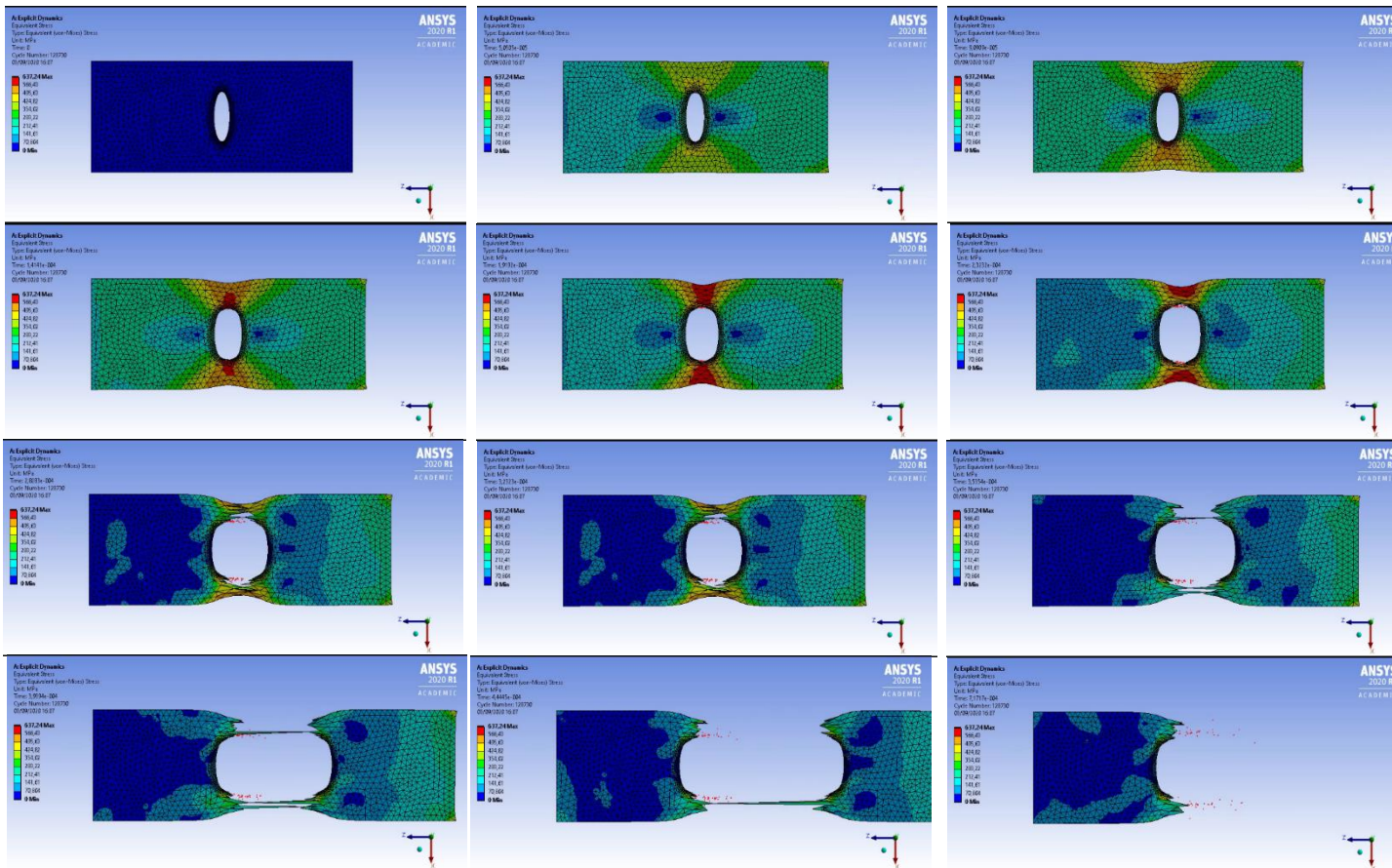
Entregas:

- ▶ Relatório
- ▶ Gravação do vídeo da simulação.



Exercício 2

Os vídeos das simulações devem ser enviados por e-mail.
No relatório deve constar a sequência conforme exemplo abaixo





Exercício 2

Tutorial para execução do exercício

Ansys -> Explicit Dynamics

Engineering Data

Engineering Data Sources

Explicit Material

selecionar o material



ANSYS explicit dynamics

The screenshot shows the ANSYS Workbench interface with the Project Schematic for an Explicit Dynamics analysis. The schematic is organized as follows:

Order	Component	Status
1	Explicit Dynamics	Active
2	Engineering Data	Completed (checkmark)
3	Geometry	Unknown (question mark)
4	Model	Unknown (question mark)
5	Setup	Unknown (question mark)
6	Solution	Unknown (question mark)
7	Results	Unknown (question mark)

The 'Explicit Dynamics' component is highlighted with a blue arrow. The 'Engineering Data' component is circled in red. The 'Geometry' component is highlighted with a blue arrow. The 'Model' component is highlighted with a blue arrow. The 'Setup' component is highlighted with a blue arrow. The 'Solution' component is highlighted with a blue arrow. The 'Results' component is highlighted with a blue arrow.

This is a zoomed-in view of the Project Schematic for an Explicit Dynamics analysis. The components are listed in a table:

Order	Component	Status
1	Explicit Dynamics	Active
2	Engineering Data	Completed (checkmark)
3	Geometry	Unknown (question mark)
4	Model	Unknown (question mark)
5	Setup	Unknown (question mark)
6	Solution	Unknown (question mark)
7	Results	Unknown (question mark)

The 'Engineering Data' component is circled in red. A blue arrow points to the 'Explicit Dynamics' component.

This is another zoomed-in view of the Project Schematic for an Explicit Dynamics analysis. The components are listed in a table:

Order	Component	Status
1	Explicit Dynamics	Active
2	Engineering Data	Completed (checkmark)
3	Geometry	Unknown (question mark)
4	Model	Unknown (question mark)
5	Setup	Unknown (question mark)
6	Solution	Unknown (question mark)
7	Results	Unknown (question mark)

The 'Engineering Data' component is highlighted with a black border. A blue arrow points to the 'Explicit Dynamics' component.

Explicit Dynamics

Explicit Dynamics



Definindo o material

The screenshot shows the ANSYS Workbench interface. In the Project Schematic, the 'Engineering Data' component is selected. The 'Engineering Data Sources' dialog is open, displaying a table of material libraries. The 'Explicit Materials' library is highlighted with a red circle, and a blue arrow points to it. Another blue arrow points to the 'Engineering Data Sources' dialog title bar.

	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			



Definindo o material

The image shows two windows from a software interface. The top window, 'Engineering Data Sources', is a table with columns A (Data Source), B (Location), C (Location), and D (Description). Row 9, 'Explicit Materials', is highlighted with a red oval and a blue arrow points to its checkbox in column B. The bottom window, 'Outline of Explicit Materials', is a table with columns A (Contents of Explicit Materials), B (Add), C (Add), and D (Source). Row 5, 'AL 1100-O', is highlighted with a red oval and a blue arrow points to its checkbox in column B. A red arrow points to the 'Adicionar' (Add) button in column C of row 5.

	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 checked="" 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)	<input checked="" type="checkbox"/>		Explicit_Materials.xml
5	AL 1100-O	<input checked="" type="checkbox"/>		Explicit_Materials.xml

The image shows a software interface with a menu bar (File, Edit, View, Tools, U, Extensions, Jobs, Help) and a toolbar. The 'Project' menu is open, and a red arrow points to it with the text 'Retornar ao projeto' (Return to project) in a dashed red box. The 'Engineering Data Sources' window is visible on the right, showing a list of material sources. The 'Project' window is also visible in the background.

Retornar ao projeto

WB Unsaved Project - Workbench

File Edit View Tools U Extensions Jobs Help

Project A2: Engineering Data

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

Engineering Data Sources

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



Exercício 2

Tutorial para execução do exercício

Ansys -> Explicit Dynamics

Engineering Data

Project

Geometry

Import geometry

Concentrado-equipe-X.IGS



Importando a geometria do corpo de prova

The image illustrates the steps to import geometry into a CAD model. On the left, a model tree shows the following structure:

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

Below the tree, the text "Explicit Dynamics" is visible. A blue arrow points from the 'Geometry' item to the context menu. The context menu is open over the 'Geometry' item and includes the following options:

- New SpaceClaim Geometry...
- New Design Explorer Geometry...
- Import Geometry
- Duplicate
- Transfer Data From New
- Transfer Data To New
- Update
- Update Upstream Components
- Refresh
- Reset
- Rename
- Properties
- Quick Help
- Add Note

A red arrow points to the 'Geometry' item in the tree, and a blue arrow points to the 'Import Geometry' option in the menu. A dashed red box labeled "Botão direito do mouse" (Right mouse button) encloses the context menu. Another dashed red box labeled "Selecionar o arquivo .IGES" (Select the .IGES file) encloses the 'Browse...' option in the sub-menu. The 'Browse...' option is also highlighted with a blue arrow. The sub-menu lists the following files:

- Flambagem-1.IGS
- Concentrador-1E.IGS
- Concentrador-3.IGS
- Concentrador-1-C.IGS



Exercício 2

Tutorial para execução do exercício

Ansys -> Explicit Dynamics

Engineering Data

Project

Geometry

Model

Gemetry

Atribuir material a geometria



Modelando

Dois clicks

Explicit Dynamics

- 1 Explicit Dynamics
- 2 Engineering Data
- 3 Geometry
- 4 Model
- 5 Setup
- 6 Solution
- 7 Results

Project*

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

Lighting

Ambient	0,1
Diffuse	0,6
Specular	1
Color	

Filter Options

Control	Enabled
---------	---------

0,000 0,010 0,020 0,030 0,040 (m)

Y
Z X

Ansys
2021 R2
STUDENT



Definido o material

The screenshot displays the Ansys Mechanical Enterprise interface. The main window shows a 3D model of a green rectangular part with a central hole. The software interface includes a top toolbar with various tools like Cut, Copy, Paste, and Solve. The left sidebar contains the Outline tree, where the 'Concentrador-exemplo-F' component is selected. The 'Engineering Data Materials' panel is open, showing a list of materials including Water Liquid, Structural Steel, Air, and AL 2024-T4. A red dashed box labeled 'Selecionar' is positioned over the material list. A red arrow points from the 'Concentrador-exemplo-F' component in the Outline to the 'AL 2024-T4' material in the list. Another red arrow points from the 'AL 2024-T4' material to the 'Assignment' field in the 'Details' panel, which is also circled in red. A blue arrow points from the 'Assignment' field back to the 'Concentrador-exemplo-F' component. A tooltip with the text 'Click to assign to the selected geometry.' is visible near the 'AL 2024-T4' material. The bottom right corner shows a 3D coordinate system with X, Y, and Z axes and a scale bar from 0,000 to 0,040 (m).



Exercício 2

Tutorial para execução do exercício

Ansyes -> Explicit Dynamics

Engeneering Data

Project

Geometry

Model

Geometry

Mesh

Gerar Mesh

Refine (na região de interesse)



Gerando a malha

The image displays two screenshots of the Ansys Mechanical Enterprise software interface, illustrating the process of generating a mesh for a 3D model of a mechanical part.

Top Screenshot: Shows the 'Sizing' menu being accessed. The 'Sizing' option is highlighted, and a tooltip explains: "Control size-related settings such as element size, number of divisions along an edge, use of sphere or body of influence, minimum size, etc. Press F1 for help."

Bottom Screenshot: Shows the 'Details of Sizing' panel. A red arrow points to the 'Geometry Selection' field, which is currently set to 'No Selection'. The panel includes the following settings:

Details of "Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	No Selection
Definition	
Suppressed	No
Type	Element Size
Element Size	Default (3,367e-003 m)
Advanced	
Default Feature Size	Default
Behavior	Soft



Gerando a malha

The image displays two screenshots of the Ansys software interface, illustrating the process of generating a mesh for a 3D model of a plate with a hole.

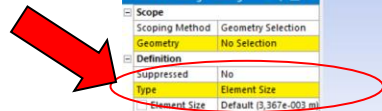
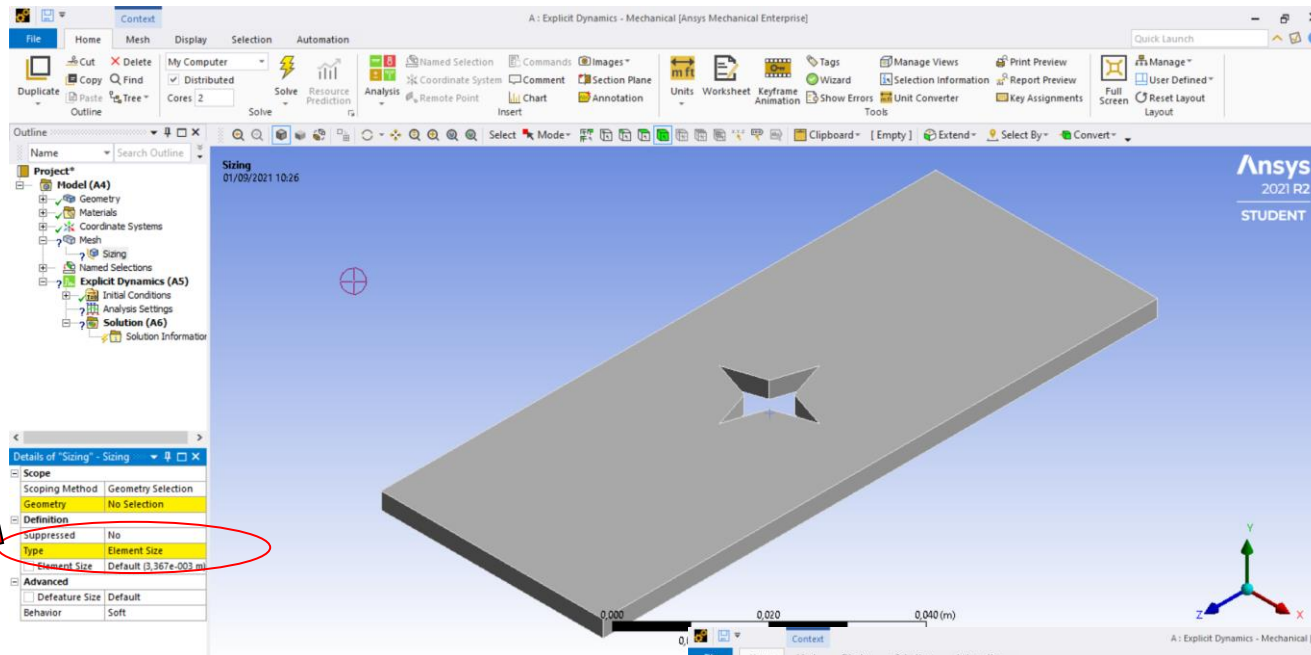
Top Screenshot: The 'Sizing' dialog box is open, showing the 'Scope' section. The 'Geometry Method' is set to 'No Selection'. A red arrow points to this setting.

Bottom Screenshot: The 'Sizing' dialog box is open, showing the 'Scope' section. The 'Geometry Method' is set to 'Geometry Selection'. The 'Apply' button is highlighted. A red arrow points to this setting.

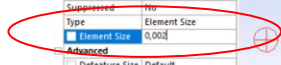
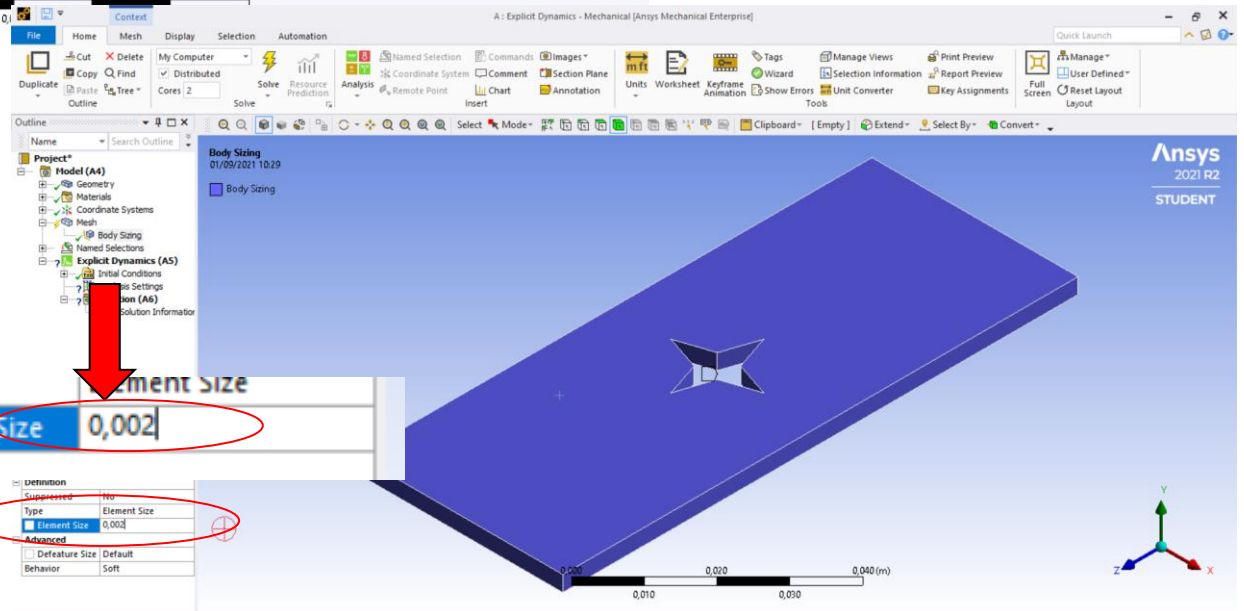
The 3D model of the plate is shown in both screenshots. In the bottom screenshot, the model is highlighted in green, indicating that the meshing process is complete or in progress.



Gerando a malha

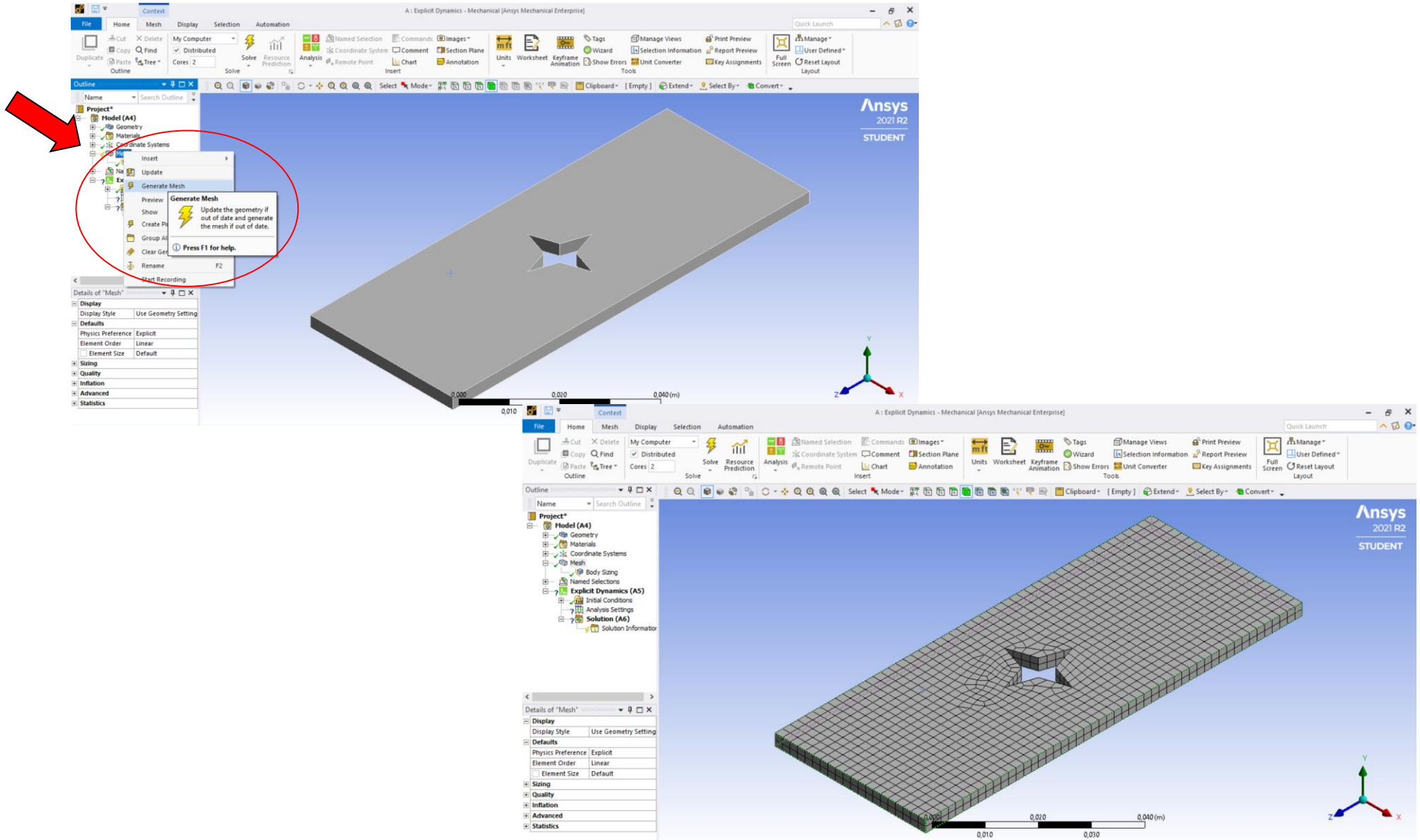


0,002m ou 2 mm





Gerando a malha





Refinando a malha

Face (Ctrl+F) Select or highlight faces on your model. Use the Ctrl button or hold the mouse button to select multiple faces. Press F1 for help.

Refinement Specify the maximum number of times you want an initial mesh to be refined. You can specify refinement controls for faces, edges, and vertices. Press F1 for help.

Details of "Refinement" - Refi -

Scoping Method	Geometry Selection
Geometry	8 Faces
Definition	
Suppressed	No
Refinement	1

Quanto maior n, maior o refinamento. Recomendo deixar em 1

Details of "Mesh" -

Display Style	Use Geometry Setting
Physics Preference	Explicit
Element Order	Linear
Element Size	Default
Sizing	
Quality	
Inflation	
Advanced	
Statistics	

Details of "Mesh" -

Display Style	Use Geometry Setting
Physics Preference	Explicit
Element Order	Linear
Element Size	Default
Sizing	
Quality	

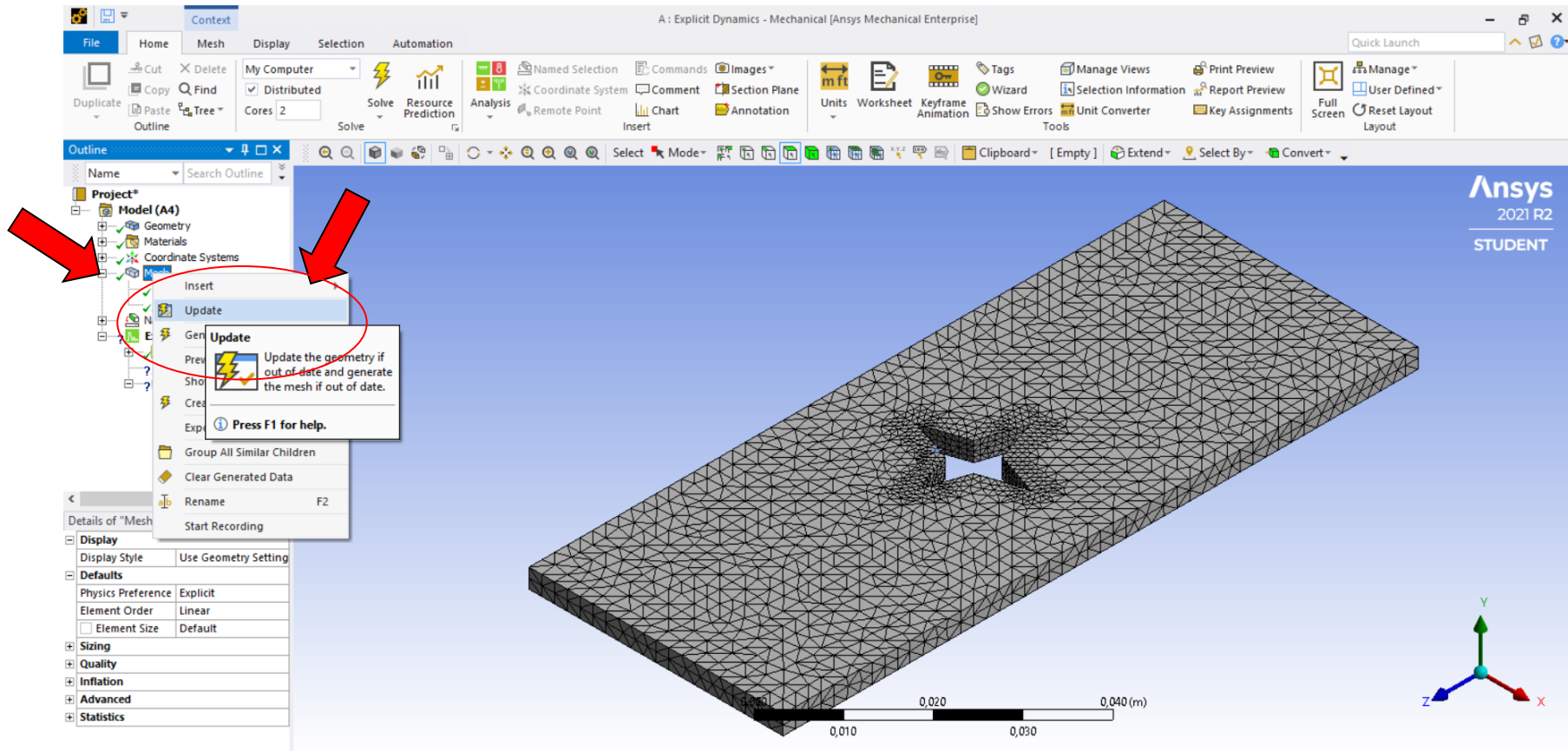
Refinement 01/09/2021 10:43

Refinement

0.010 0.010 0.040 (m)



Refinando a malha





Definição das condições de contorno

Carregamento

The image shows the Ansys Mechanical Enterprise software interface. The main window displays a 3D model of a mechanical part. The 'Details of Force' dialog box is open, showing the following settings:

Details of "Force"	
Scoping Method	Geometry Selection
Geometry	Geometry Selection
Named Selection	Named Selection
Definition	
Type	Force
Define By	Vector
Magnitude	<input type="checkbox"/> 0, N (step applied)
Direction	Click to Define
Suppressed	No

The 'Details of Force' dialog box is also open, showing the following settings:

Details of "Force"	
Scoping Method	Named Selection
Named Selection	Named Selection
Definition	Fix Load
Type	Vector
Define By	Vector
Magnitude	<input type="checkbox"/> 0, N (step applied)
Direction	Click to Define
Suppressed	No

The 'Details of Force' dialog box is also open, showing the following settings:

Details of "Force"	
Scope	
Scoping Method	Named Selection
Named Selection	Load
Definition	
Type	Force
Define By	Component
Coordinate System	Components
<input type="checkbox"/> X Component	0, N (step applied)
<input type="checkbox"/> Y Component	0, N (step applied)
<input type="checkbox"/> Z Component	-10000 N (step applied)
Suppressed	No



Exercício 2

Tutorial para execução do exercício

Ansyc -> Explicit Dynamics

Engeneering Data

Project

Geometry

Model

Geometry

Mesh

Explicit Dynamics

Fix (definir face de restrição)

Load (definir face de carregamento)



Definição das áreas de interesse

Fixação (Fix)

The image shows a sequence of four screenshots from the Ansys 2021 R2 Student software interface, illustrating the process of defining a 'Fix' boundary condition on a 3D model. Red arrows and circles highlight key actions:

- Top Screenshot:** Shows the software toolbar with the 'Fix' icon (a green square with a white 'F') highlighted by a red arrow.
- Second Screenshot:** Shows the 3D model of a rectangular plate. The bottom edge is highlighted in green, and a red circle is drawn around it. A red arrow points to this edge.
- Third Screenshot:** Shows the 'Create Named Selection' dialog box. The 'Name' field is set to 'Fix'. A red circle is drawn around the 'Fix' text in the dialog box, with a red arrow pointing to it.
- Bottom Screenshot:** Shows the 3D model with the bottom edge highlighted in green. A red circle is drawn around the highlighted edge, with a red arrow pointing to it.



Definição das condições de contorno

Fixação

The screenshot shows the Ansys Mechanical Enterprise interface. The main window displays a 3D model of a rectangular plate with a hole. A context menu is open over the model, and a 'Fixed Support' dialog box is visible. The dialog box shows the 'Scope' section with 'Scoping Method' set to 'Named Selection' and 'Definition' set to 'Named Selection'. A second dialog box shows the 'Definition' section with 'Fix' selected, which is circled in red.

Details of "Fixed Support"	
Scope	
Scoping Method	Named Selection
Named Selection	Geometry Selection
Definition	Named Selection
Type	Fixed Support
Suppressed	No

Details of "Fixed Support"	
Scope	
Scoping Method	Named Selection
Named Selection	
Definition	Fix
Type	Load
	Fixed Support
Suppressed	No



Definição das condições de contorno

Fixação

The image shows a screenshot of the Ansys software interface. The main window displays a 3D model of a mechanical part with a fixed support boundary condition applied to its bottom edge. The software interface includes a toolbar with various tools like 'Solve', 'Resource Prediction', 'Analysis', 'Coordinate System', 'Comment', 'Section Plane', 'Units', 'Worksheet', 'Keyframe Animation', 'Tags', 'Wizard', 'Manage Views', 'Print Preview', 'Report Preview', 'Full Screen', and 'User Defined'. The 'Explicit Dynamics (A5)' tree on the left shows the analysis setup, with 'Fixed Support' highlighted. A red arrow points to the 'Fixed Support' icon in the tree. A 'Details of "Fixed Support"' panel is open, showing the following information:

Details of "Fixed Support"	
Scope	
Scoping Method	Named Selection
Named Selection	Fix
Definition	
Type	Fixed Support
Suppressed	No

The 3D model shows a coordinate system with X, Y, and Z axes. The bottom edge of the part is highlighted in blue, indicating the fixed support. The dimensions of the part are shown as 0,000, 0,015, 0,030, and 0,060 (m).



Definição das áreas de interesse

Carregamento (Load)

The image displays two screenshots of the Ansys Mechanical software interface, illustrating the steps to define a named selection for a load.

Top Screenshot: Shows the software interface with a 3D model of a part. A red arrow points to the **Named Selection** icon in the toolbar. Another red arrow points to the **Create Named Selection...** dialog box, which is open. The dialog box contains the following options:

- Create a Named Selection for the selected geometry entries in the graphical interface (Bodies, faces, etc.). You can specify a name for the selection and you can specify criteria based on the selected geometry.
- Selection Name
- Apply selected geometry
- Apply geometry items of same:
 - Size
 - Type
 - Location X
 - Location Y
 - Location Z
- Apply To Corresponding Mesh Nodes

Bottom Screenshot: Shows the same software interface with the 3D model. A red arrow points to the **Named Selection** icon in the toolbar. Another red arrow points to the **Create Named Selection...** dialog box, which is open. The dialog box contains the following options:

- Create a Named Selection for the selected geometry entries in the graphical interface (Bodies, faces, etc.). You can specify a name for the selection and you can specify criteria based on the selected geometry.
- Selection Name
- Apply selected geometry
- Apply geometry items of same:
 - Size
 - Type
 - Location X
 - Location Y
 - Location Z
- Apply To Corresponding Mesh Nodes



Definição das condições de contorno

Região de Carregamento

The image shows a screenshot of the Ansys software interface. The main window displays a 3D model of a mechanical part with a force boundary condition applied. The force is defined as a component force of -10000 N in the Z direction. The software interface includes a toolbar with various tools, a command window, and a details panel for the force boundary condition.

Explicit Dynamics (A5) Tree:

- Initial Conditions
- Analysis Settings
- Force
- Solution (A6)
- Solution Information

Details of "Force" Panel:

Details of "Force"	
Scoping Method	Named Selection
Named Selection	Load
Definition	
Type	Force
Define By	Component
Coordinate System	Components
<input type="checkbox"/> X Component	0, N (step applied)
<input type="checkbox"/> Y Component	0, N (step applied)
<input checked="" type="checkbox"/> Z Component	-10000 N (step applied)
Suppressed	No



Exercício 2

Tutorial para execução do exercício

Ansyc -> Explicit Dynamics

Engeneering Data

Project

Geometry

Model

Geometry

Mesh

Explicit Dynamics

Fix (definir face de restrição)

Load (definir face de carregamento)

Analysis Settings



Definição dos parâmetros da solução

The screenshot displays the Ansys Mechanical software interface. The 'Analysis Settings' dialog box is open, showing the 'Analysis Settings Preference' section. The 'End Time' parameter is highlighted in yellow and circled in red, with a red arrow pointing to it. The value '0,001' is also circled in red and has a red arrow pointing to it. The background shows a 3D model of a mechanical part and the software's menu and toolbar.

Analysis Settings Preference	
Type	Program Controlled
Step Controls	
Number Of Steps	1
Current Step Number	1
Load Step Type	Explicit Time Integrat...
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
Maximum Time Step	Program Controlled
Time Step Safety Factor	0,9



Definição dos parâmetros da solução

The screenshot displays the Ansys Mechanical Enterprise interface. The 'Details of "Analysis Settings"' dialog box is open, showing the 'Erosion Controls' section. The following table represents the settings shown in the dialog:

Setting	Value
Upper Y Face	Flow Out
Upper Z Face	Flow Out
Euler Tracking	By Body
Damping Controls	
Erosion Controls	
On Geometric Strain Limit	Yes
Geometric Strain Limit	1,5
On Material Failure	No
On Minimum Element Time Step	No
Retain Inertia of Eroded Material	Yes
Output Controls	
Analysis Data Management	
Solver Files Directory	C:\Users\risto\AppData...
Scratch Solver Files Directory	

Red arrows in the image point to the 'On Material Failure' and 'Retain Inertia of Eroded Material' settings, indicating they are the focus of the configuration.



Exercício 2

Tutorial para execução do exercício

Ansyes -> Explicit Dynamics

Engineering Data

Project

Geometry

SOLVE

Geometry

Mesh

Explicit Dynamics

Fix (definir face de restrição)

Load (definir face de carregamento)

Analysis Settings



Solve

The screenshot shows the Ansys Mechanical Enterprise software interface. The main window displays a 3D model of a rectangular plate with a force applied. The 'Solve' button is highlighted in the 'Named Selections' panel. A tooltip for the 'Solve' button is visible, stating: "Start the solution process using the current solve process settings. Press F1 for help." The interface includes a top toolbar, a left-hand 'Outline' tree, and a bottom 'Details' panel for the selected 'Explicit Dynamics (A5)' solution.

Named Selections

- Explicit Dynamics (A5)
- Initial Co
- Analysis
- Force
- Fixed Su
- Solutio

Solve

Start the solution process using the current solve process settings.

Press F1 for help.

Details of "Explicit Dynamic"

Definition

Physics Type	Structural
Analysis Type	Explicit D...
Solver Target	AUTODYN
Options	
Environment Temperature	22, °C
Generate Input Only	No



Exercício 2

Tutorial para execução do exercício

Ansys -> Explicit Dynamics

Engineering Data

Project

Geometry

Resultados

Geometry

Mesh

Explicit Dynamics

Fix (definir face de restrição)

Load (definir face de carregamento)

Analysis Settings



Resultados

The screenshot displays the Ansys Mechanical Enterprise interface for an explicit dynamics simulation. The main window shows a 3D model of a beam with a hole. The software version is 2021 R2, and the user is identified as a student. The interface includes a toolbar with various tools like Cut, Copy, Find, and Solve. The Outline panel on the left shows the project structure, with the 'Solution (A6)' node highlighted. The 'Deformation' menu is open, showing options like Total, Strain, Stress, and Linearized Stress. A tooltip for the 'Total' option states: "Insert a Total Deformation object. This result provides the magnitude of displacements on nodes." The 3D view shows the beam with a hole, and dimensions are indicated: 0,015, 0,030, 0,045, and 0,060 (m). A coordinate system (X, Y, Z) is visible in the bottom right.



Resultados

The screenshot displays the Ansys Mechanical Enterprise software interface. The main window shows a 3D model of a mechanical part, likely a bracket or a similar component, with a coordinate system (X, Y, Z) and a scale bar indicating dimensions of 0,030, 0,045, and 0,060 (m). The software is running an explicit dynamics simulation, as indicated by the title bar and the 'Explicit Dynamics' label in the Outline.

The Outline panel on the left shows the project structure, with the 'Solution' branch selected. The 'Stress' > 'Equivalent (von-Mises)' path is highlighted with red arrows and circles. A tooltip for 'Equivalent (von-Mises)' is visible, explaining its purpose: "Insert an Equivalent (von-Mises) stress object to determine the overall stress at each element based on the Von-Mises criterion." The tooltip also includes a note: "Press F1 for help."

The main window displays the 'Solution' results, showing the 'Equivalent (von-Mises)' stress distribution. The software version is Ansys 2021 R2, and the user is identified as a STUDENT.



Resultados

The screenshot displays the Ansys Mechanical Enterprise software interface. The main window shows a 3D model of a mechanical part with a coordinate system (X, Y, Z) and a scale bar indicating dimensions of 0,045 and 0,060 (m). The software title bar reads "A: Explicit Dynamics - Mechanical [Ansys Mechanical Enterprise]".

The **Outline** panel on the left shows the project structure:

- Project*
- Model (A4)
 - Geometry
 - Materials
 - Coordinate Systems
 - Mesh
 - Body Sizing
 - Refinement
 - Named Selections
 - Explicit Dynamics (A5)
 - Initial Conditions
 - Analysis Settings
 - Force
 - Fixed Support
 - Solution (A6)
 - Solution Information
 - Total Deformation
 - Equivalent Stress

The **Details of "Solution (A6)"** panel at the bottom left shows the following information:

Information	Post-processing ...
Status	
Post Processing	
Beam Section Results	No

The **Explicit Dynamics (A5)** panel in the center shows the following settings:

- Initial Conditions
- Analysis Settings
- Force
- Fixed Support
- Solution (A6)
- Solution Information
- Total Deformation
- Equivalent Stress

The **Solution (A6)** panel at the bottom of the center shows the following results:

- Solution Information
- Total Deformation
- Equivalent Stress



Resultados

The screenshot displays the Ansys Mechanical Enterprise interface for an explicit dynamics analysis. The software window title is "A: Explicit Dynamics - Mechanical [Ansys Mechanical Enterprise]". The interface includes a menu bar (File, Home, Solution, Display, Selection, Automation), a toolbar with various tools, and an Outline tree on the left. The Outline tree shows a project structure with folders for "Model (A4)", "Explicit Dynamics (A5)", and "Solution (A6)". The "Explicit Dynamics (A5)" folder is expanded, showing sub-items like "Initial Conditions", "Analysis Settings", "Force", and "Fixed Support". The "Solution (A6)" folder is also expanded, showing "Solution Information", "Total Deformation", and "Equivalent Stress". A red dashed box highlights the "Explicit Dynamics (A5)" folder. A red arrow points from this folder to a context menu that is open over the "Solution (A6)" folder. The context menu has "Evaluate All Results" selected, and a tooltip for this option is visible, stating "Evaluate all results. Press F1 for help." Another red arrow points from the context menu to the "Solution (A6)" folder in the Outline tree, which is circled in red. The "Solution (A6)" folder is also circled in red, and its sub-items are circled in red. The main 3D view shows a grey rectangular block with a blue star-shaped hole in the center, set against a blue background. The Ansys logo and "2021 R2 STUDENT" are visible in the top right corner of the software window.



Resultados

A: Explicit Dynamics
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: Pa
Time: 2,6317e-004
Cycle Number: 27211
02/09/2021 14:15

6,6662e8 Max
5,9255e8
5,1849e8
4,4441e8
3,7034e8
2,9627e8
2,2221e8
1,4814e8
7,4069e7
0 Min

Grava o vídeo

Inicia o vídeo

Details of "Equivalent Stress"

Scope	Geometry Selection
Scoping Method	Geometry Selection
Geometry	All Bodies

Definition

Type	Equivalent (von-Mises) Stress
By	Time
Display Time	8,5003e-004 s
Calculate Time History	Yes
Identifier	
Suppressed	No

Integration Point Results

Time [s]	Minimum [Pa]	Maximum [Pa]	Average [Pa]
1	1,1755e-038	0,	0,
2	5,0019e-005	5,0183e+007	6,6662e+008
3	1,0003e-004	2,7075e+007	6,0432e+008
4	1,5004e-004	1,3355e+007	6,1521e+008
5	2,0001e-004	1,2938e+007	6,1561e+008
6	2,5003e-004	8,7193e+006	6,2282e+008
7	3,0002e-004	1,7511e+007	5,5227e+008



Exercício 2

Vídeos exemplo

 <https://www.youtube.com/watch?v=zPI3-dyvpEY&t=32s>

 <https://www.youtube.com/watch?v=vnpq5zzOS48>



FIM