

Dynamická úloha, explicitní děj

Ing. Petr Lehner

Co se dozvíme a naučíme?

Základní popis dynamické úlohy.

Rozdíl mezi implicitní a explicitní analýzou.

Provázání statického a dynamického děje.

Vysoce zjednodušená analýza pomocí explicitního výpočtu.

Dynamická úloha

Na rozdíl od statiky, bere do úvahy navíc také pohyb hmotného tělesa, tedy působení setrvačných a tlumících sil.

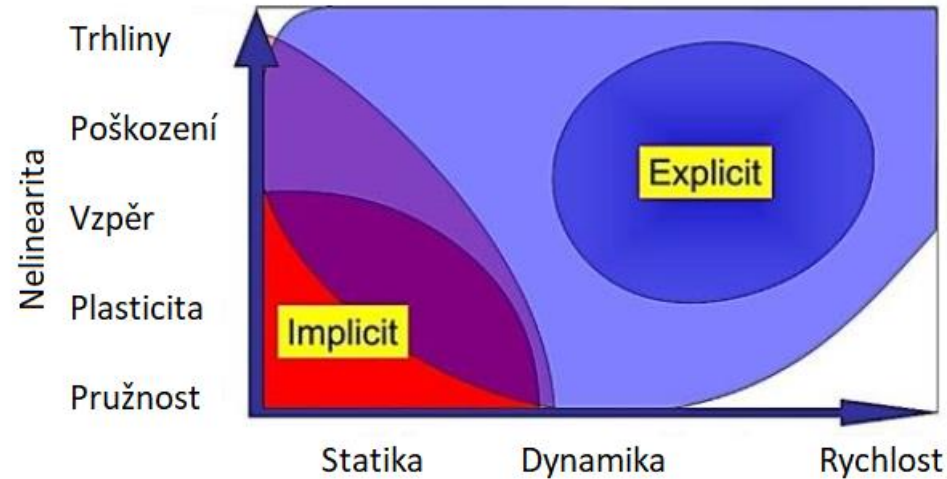
Explicitní vs implicitní řešení

Rozdíl je v přístupu k časovému přírůstku.

V implicitní analýze musí každý časový přírůstek konvergovat, ale můžete nastavit poměrně dlouhé časové přírůstky.

Naopak, explicitní analýza nemusí

konvergovat každý přírůstek, ale časové přírůstky musí být pro přesnost řešení super malé.



Laicky řečeno: pokud je děj rychlý (náraz, výbuch, rychlá deformace), lze na něj aplikovat explicit (který je náročný na výpočetní výkon). Pokud je děj zdlouhavější využije se implicit.

Příklad

Materiál: borovicové dřevo (pine wood)

$E = 9\,300\text{ MPa}$

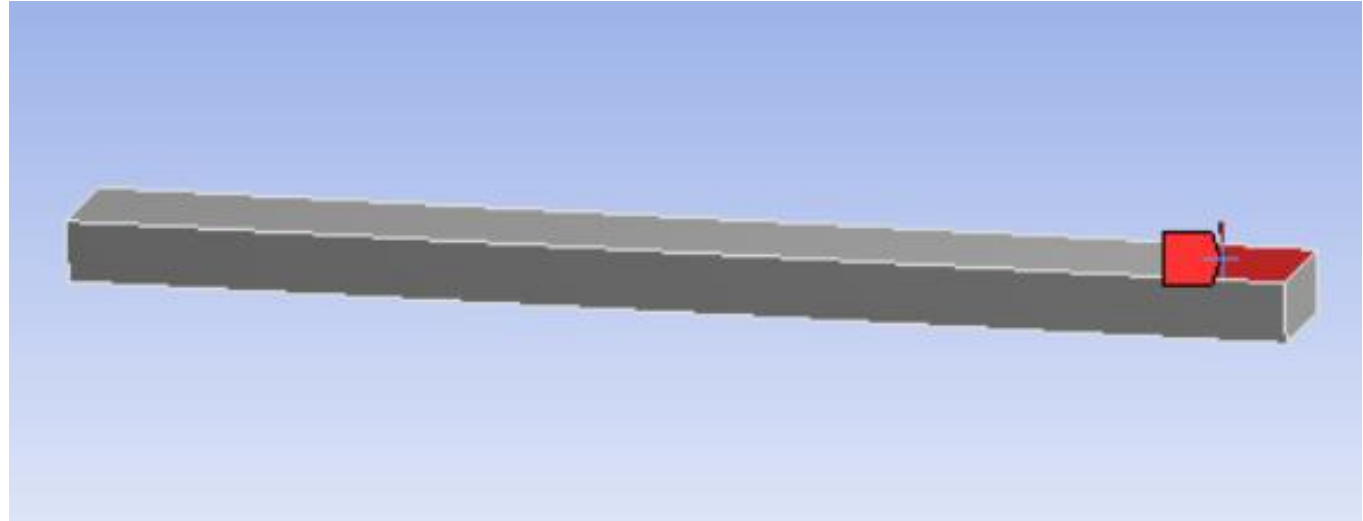
$\mu = 0.374$

$\rho = 487\text{ kg/m}^3$

Obdélník

$a = 0.1\text{ m}$

$b = 0.05\text{ m}$



Délka prutu: 1 m

Zatížení na koncové ploše 5 kg – 0,1 x 0,1 m

$5 \times 9,81 / (0,1 \times 0,1) = 4905\text{ Pa}$

Ansys Workbench

Unsaved Project - Workbench

File View Tools Units Extensions Jobs Help

Project A2,B2:Engineering Data

Import... Reconnect Refresh Project Update Project ACT Start Page

Toolbox Project Schematic Properties of Schematic A3: Geometry

Fluid Flow (CFX)
 Fluid Flow (Fluent)
 Fluid Flow (Polyflow)
 Harmonic Acoustics
 Harmonic Response
 Hydrodynamic Diffraction
 Hydrodynamic Response
 IC Engine (Fluent)
 Magnetostatic
 Modal
 Modal Acoustics
 Random Vibration
 Response Spectrum
 Rigid Dynamics
 Static Acoustics
 Static Structural
 Steady-State Thermal
 Thermal-Electric
 Topology Optimization
 Transient Structural
 Transient Thermal
 Turbomachinery Fluid Flow

Component Systems
 ACP (Post)
 ACP (Pre)
 Autodyn
 BladeGen
 CFX
 Engineering Data
 External Data
 External Model
 Fluent
 Fluent (with Fluent Meshing)
 Forte
 Geometry
 GRANTA MI
 ICEM CFD
 Trans

View All / Customize...

Static Structural

Explicit Dynamics

Engineering Data Source

	A	B
1	Property	Value
2	General	
3	Component ID	Geometry
4	Directory Name	SYS
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	Basic Geometry Options	
10	Solid Bodies	<input checked="" type="checkbox"/>
11	Surface Bodies	<input checked="" type="checkbox"/>
12	Line Bodies	<input type="checkbox"/>
13	Parameters	Independent
14	Parameter Key	ANS;DS
15	Attributes	
16	Named Selections	<input type="checkbox"/>
17	Material Properties	<input type="checkbox"/>
18	Advanced Geometry Options	
19	Analysis Type	3D
20	Use Associativity	<input checked="" type="checkbox"/>
21	Import Coordinate Systems	<input type="checkbox"/>
22	Import Work Points	<input type="checkbox"/>
23	Reader Mode Saves Updated File	<input type="checkbox"/>
24	Import Using Instances	<input checked="" type="checkbox"/>
25	Smart CAD Update	<input checked="" type="checkbox"/>
26	Compare Parts On Update	No
27	Enclosure and Symmetry Processing	<input checked="" type="checkbox"/>
28	Decompose Disjoint Geometry	<input checked="" type="checkbox"/>
29	Clean Geometry On Import	<input type="checkbox"/>
30	Stitch Surfaces On Import	None
31	Mixed Import Resolution	None

Starting SpaceClaim...

Job Monitor... No DPS Connection Show Progress Show 0 Messages

Engineering data – vlastnosti materiálu

Unsaved Project - Workbench

File Edit View Tools Units Extensions Jobs Help

Project A2,B2:Engineering Data

Filter Engineering Data Engineering Data Sources

Toolbox

- Field Variables
 - Temperature
 - Frequency
 - Coordinate X
 - Coordinate Y
 - Coordinate Z
 - Shear Angle
 - Degradation Factor
 - Create Field Variable ...
- Physical Properties
 - Density
 - Isotropic Secant Coefficient of T
 - Orthotropic Secant Coefficient of
 - Isotropic Instantaneous Coeffie
 - Orthotropic Instantaneous Coeff
 - Melting Temperature
 - Linear Elastic
 - Hyperelastic Experimental Data
 - Hyperelastic
 - Chaboche Test Data
 - Plasticity
 - Bilinear Isotropic Hardening
 - Multilinear Isotropic Hardening
 - Nonlinear Isotropic Hardening Pc
 - Nonlinear Isotropic Hardening Vc
 - Bilinear Kinematic Hardening
 - Multilinear Kinematic Hardening
 - Chaboche Kinematic Hardening
 - Chaboche Kinematic Hardening v
 - Anand Viscoplasticity
 - Exponential Visco-Hardening (EV
 - Perzyna Viscoplasticity
 - Peirce Viscoplasticity
 - Gurson Model
 - Hill Yield Criterion
 - Johnson Cook Strength
 - Cowner Svmonds Strength

View All / Customize...

Engineering Data Sources

	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 full product with broader coverage of material data (e.g. linear, non-linear, temperature dependant, fatigue etc.) which includes more than 700 material datasheets.

Outline of Granta Design Sample Materials

	A	B	C	D	E
1	Contents of Granta Design Sample Materials	Add	source		Description
64	PHENOLIC RESIN (PP)				Additional data and information available through the Granta website . Granta provides no warranty for the accuracy of the data.
65	Pine wood				Pine (pinus spp.), longitudinal Sample materials data from Granta Design. Additional data and information available through the Granta website . Granta provides no warranty for the accuracy of the data.
66	Plywood				Plywood, 5 ply beech, parallel to face sheet Sample materials data from Granta Design. Additional data and information available through the Granta website . Granta provides no warranty for the accuracy of the data.
67	Polycarbonate (PC)				Polycarbonate (PC), low viscosity Sample materials data from Granta Design. Additional data and information available

Properties of Outline Row 65: Pine wood

	A	B	C
1	Property	Value	Unit
2	Density	487	kg m^-3
3	Isotropic Secant Coefficient of Thermal Expansion		
4	Coefficient of Thermal Expansion	4,69E-06	C^-1
5	Isotropic Elasticity		

Table of Properties Row 2: Density

	A
1	Density (kg m^-3)
2	487

Chart: No data

Ready Job Monitor... No DPS Connection Show Progress Show 0 Messages

Geometrie – Space Claim

The image shows the ANSYS SpaceClaim 2020 R1 interface. The main workspace displays a 2D sketch of a rectangle on a blue grid. The rectangle is defined by a green outline. A dimension line indicates a length of 100mm. A callout box labeled "Obdélník" points to the rectangle. Another callout box labeled "50 x 100" is positioned near the rectangle. A third callout box labeled "Změna kreslicí plochy" points to the "Sketch" button in the bottom toolbar. The ribbon at the top includes tabs for File, Design, Display, Assembly, Measure, Facets, Repair, Prepare, Workbench, Detail, Sheet Metal, Tools, and KeyShot. The left sidebar shows the Structure tree with "Design1*" and "Curves". The Options - Sketch panel is open, showing "Rectangle" selected with "Define rectangle from center" and "Snap to grid" checked. The Properties panel is empty. The bottom status bar shows "Snapping to Grid" and coordinates "x=-2,000 y=-120,000".

Obdélník

50 x 100

100mm

Změna kreslicí plochy

Geometrie – tvorba objemu

A:Static Structural - SYS - SpaceClaim

File Design Display Assembly Measure Facets Repair Prepare Workbench Detail Sheet Metal Tools KeyShot

Paste Plan View Pan Zoom Orient Sketch Mode Select Pull Move Fill Combine Split Body Split Project Intersect Create Shell Offset Mirror Body

Structure

- ✓ SYS*
- ✓ Solid

Options - Pull

General

+ Add - Cut No merge

Properties

ANSYS 2020 R1 ACADEMIC

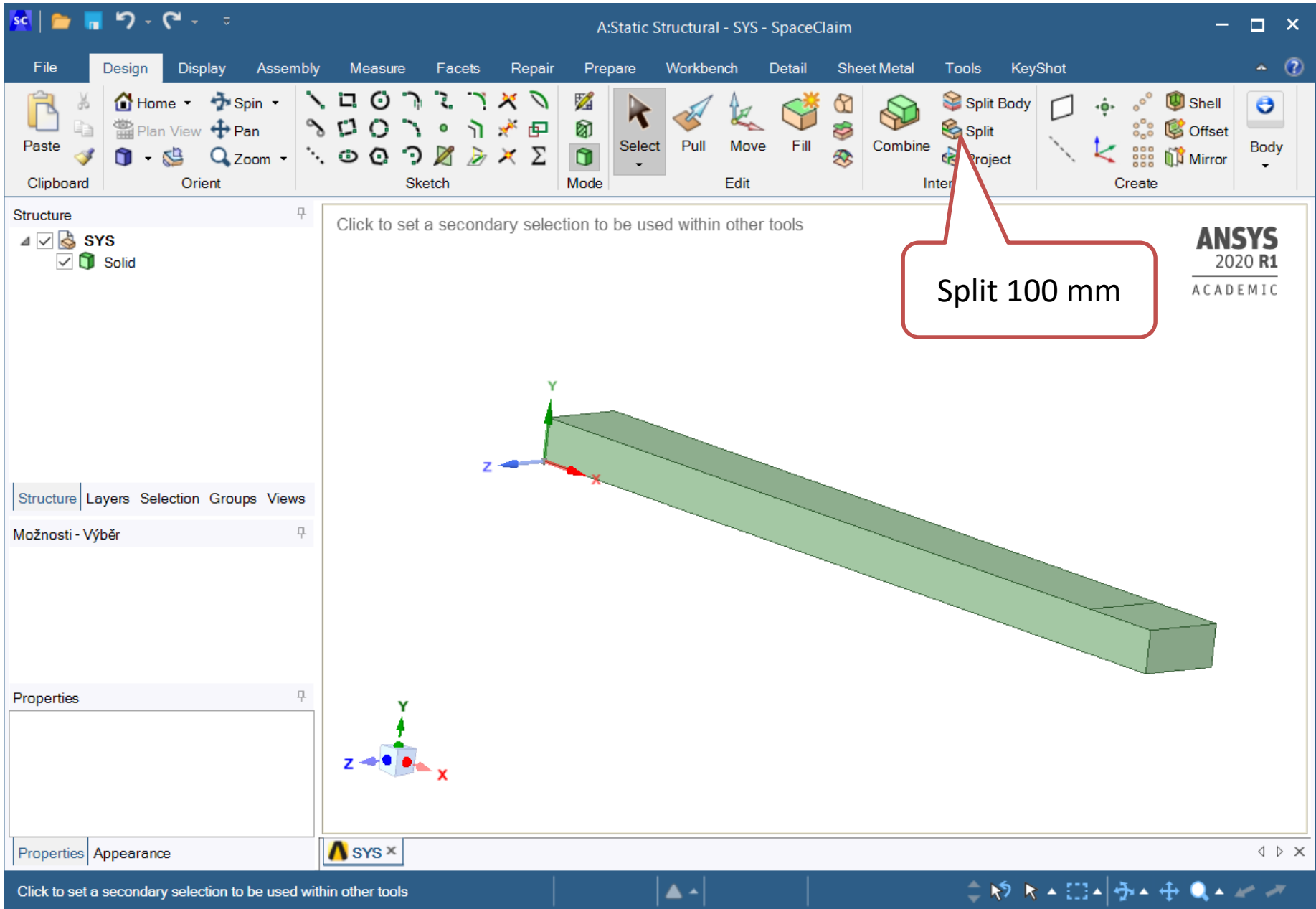
Select and drag a face to offset it. Select and drag an edge to round it.

Protažení 1 m

Face

Select and drag a face to offset it. Select and drag an edge to round it.

Geometrie – tvorba plochy pro tlak



Model – přiřazení materiálu

The screenshot displays the ANSYS 2020 R1 Academic software interface. The main window shows a 3D model of a green rectangular block. A red callout box points to the block with the text "Pine wood". The software interface includes a top menu bar with options like File, Home, Geometry, Display, Selection, and Automation. The left sidebar shows the Outline tree with a hierarchy: Project* > Model (A4, B4) > Geometry > SYS\Solid. The central panel shows the Details of "SYS\Solid" with various properties. The right panel shows the ANSYS logo and version information. The bottom status bar indicates "1 Message" and "No Selection".

Details of "SYS\Solid"

Graphics Properties	
Definition	
<input type="checkbox"/> Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate Sy...
Reference Temperature	By Environment
Treatment	None
Reference Frame	Lagrangian
Material	
Assignment	Pine wood
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Properties	
Statistics	
CAD Attributes	
PartTolerance:	0,00000001
Color:	143.175.143

ANSYS 2020 R1 ACADEMIC

0,00 150,00 300,00 (mm)

Messages: Text
Warning: One or more objects may have lost some scoping attachments during the geometry update

1 Message No Selection Metric (mm, t, N, s, mV, mA) Degrees rad/s Celsius

Mesh – default size a explicit

The screenshot displays the ANSYS 2020 R1 Academic software interface. The main window shows a 3D model of a rectangular block with a mesh applied. The mesh is composed of small, uniform elements. A scale bar at the bottom right indicates a length of 300.00 mm, with a midpoint at 150.00 mm. The coordinate system (X, Y, Z) is visible in the bottom right corner.

The **Details of "Mesh"** panel is open, showing the following settings:

Property	Value
Display Style	Use Geometry Setting
Physics Preference	Explicit
Element Order	Linear
Element Size	25, mm

Two callout boxes highlight the **Explicit** physics preference and the **25 mm** element size. A context menu is open over the **Mesh** icon in the Outline, with the **Generate Mesh** option selected. A tooltip for **Generate Mesh** is displayed, stating: "Update the geometry if out of date and generate the mesh if out of date. Press F1 for help."

The Messages panel at the bottom shows a warning: "One or more objects may have lost some scoping attachments during the geometry update."

At the bottom of the interface, the status bar displays: "Update the geometry if out of date and generate the mesh if out of date." and "1 Message No Selection Metric (mm, t, N, s, mV, mA) Degrees rad/s Celsius".

Static Structural - Fixed Support

The screenshot displays the ANSYS Academic Research Mechanical and CFD software interface. The main window shows a 3D model of a cylindrical part with a fixed support applied to one end. The software is running a Static Structural analysis.

Context: Multiple Systems - Mechanical [ANSYS Academic Research Mechanical and CFD - ANSYS Academic Research Mechanical and CFD]

File Menu: Duplicate, Cut, Copy, Paste, Find, Tree, Outline

Environment: My Computer, Distributed, Cores 2, Solve

Automation: Named Selection, Commands, Images, Coordinate System, Comment, Section Plane, Remote Point, Chart, Annotation, Tools, Layout

Outline: Project*, Model (A4, B4), Geometry, SYS\Solid, Materials, Pine wood, Coordinate Systems, Mesh, Static Structural (A5), Analysis Settings, Fixed Support, Solution (A6), Explicit Dynamics (B5), Initial Conditions, Analysis Settings, Solution (B6)

Details of "Fixed Support":

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Fixed Support
Suppressed	No

A: Static Structural
Fixed Support
Time: 2,e-002 s
30.04.2021 14:57

ANSYS 2020 R1 ACADEMIC

Fixed Support

0,00 100,00 200,00 (mm)

Graph: 2,e-2

Messages | Graph

1 Message No Selection Metric (mm, t, N, s, mV, mA) Degrees rad/s Celsius

Static Structural – pressure – na ploše

The screenshot displays the ANSYS 2020 R1 Academic software interface. The main window shows a 3D model of a rectangular block with a red square area on its top surface, representing a pressure load. The software title bar indicates the project is "Multiple Systems - Mechanical [ANSYS Academic Research Mechanical and CFD - ANSYS Academic Research Mechanical and CFD]".

The **Details of "Pressure"** panel shows the following configuration:

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Pressure
Define By	Normal To
Applied By	Surface Effect
Magnitude	4,905e-003 MPa (ramped)
Suppressed	No

The **Graph** panel shows a plot of the pressure magnitude over time. The y-axis ranges from 0 to 4,905e-3 MPa. The x-axis represents time in seconds, with a scale from 0 to 2e-2. A red dashed line shows the pressure increasing linearly from 0 to 4,905e-3 MPa over the 2e-2 second interval.

Tabular Data			
Steps	Time [s]	Pressure [MPa]	
1	0,	= 0,	
2	2,e-002	4,905e-003	
*			

The status bar at the bottom indicates "Ready" and "1 Message". The units are set to Metric (mm, t, N, s, mV, mA), Degrees, rad/s, and Celsius.

Nastavení Total deformation a Equivalent Stress

The screenshot displays the ANSYS 2020 R1 Academic software interface. The main window shows a 3D model of a long, thin rectangular bar. The software is in the 'Solution' phase, and the 'Details of "Solution (B6)"' panel is open, showing 'Status: Solve Required' and 'Post Processing: Beam Section Results: No'. The 'Outline' panel on the left shows the project structure, with 'Explicit Dynamics (B5)' selected. A red callout box with the word 'definice' (definition) points to the 'Explicit Dynamics (B5)' entry in the outline. The 'Graph' panel at the bottom shows a plot of time in seconds, with a scale from 0 to 5e-2. The 'Tabular Data' panel on the right shows a table of time values in seconds.

Outline:

- Project*
- Model (A4, B4)
 - Geometry
 - SYS\Solid
 - Materials
 - Pine wood
 - Coordinate Systems
 - Mesh
 - Static Structural (A5)
 - Analysis Settings
 - Fixed Support
 - Pressure
 - Solution (A6)
 - Solution Information
 - Total Deformation
 - Equivalent Stress
 - Explicit Dynamics (B5)
 - Initial Conditions
 - Analysis Settings
 - Support
 - Solution (B6)

Details of "Solution (B6)"

Information	
Status	Solve Required
Post Processing	
Beam Section Results	No

B: Explicit Dynamics
Solution
Time: 5,e-002 s
30.04.2021 15:07

ANSYS 2020 R1 ACADEMIC

0,00 400,00 (mm)
200,00

Graph

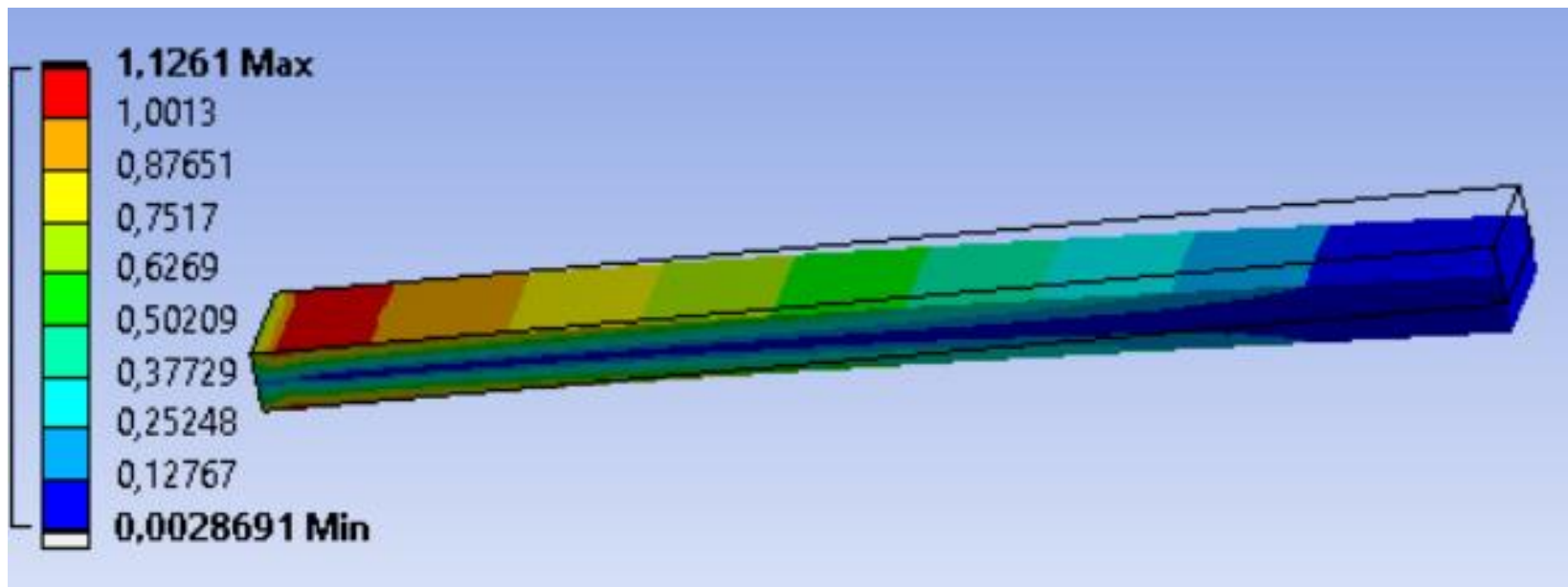
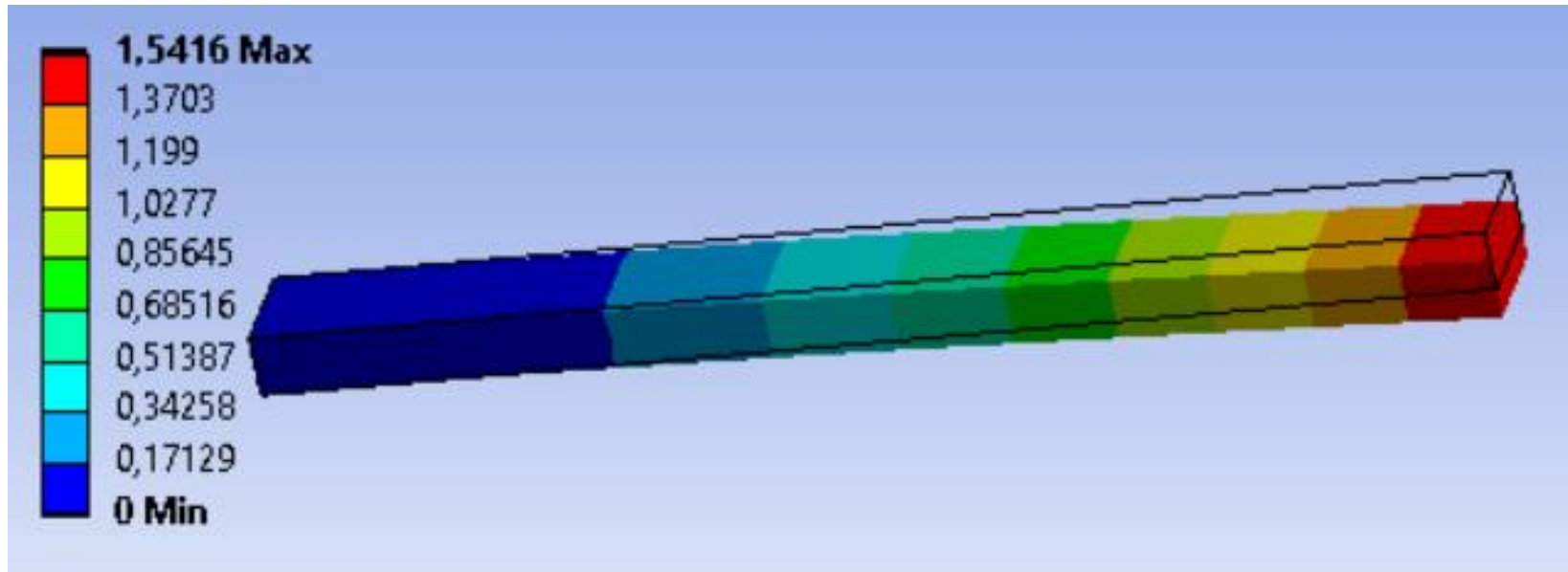
5,e-2
0, 1,e-2 2,e-2 3,e-2 5,e-2
[s]

Tabular Data

	Time [s]
1	1,1755e-038
2	5,9519e-003
3	1,1901e-002
4	1,7852e-002
5	2,0002e-002
6	2,1001e-002
7	2,2003e-002
8	2,3002e-002

Messages Graph (43%) Solving the mathematical model... 1 Message No Selection Metric (mm, t, N, s, mV, mA) Degrees rad/s Celsius

Výsledky – deformace (mm) a napětí (MPa)



Nastavení Explicit Dynamics

Analysis Settings Preference

Type	Program ...
Number Of Steps	1
Current Step Number	1
Load Step Type	Explicit Ti...
End Time	2, e-002 s
Resume From Cycle	0
Maximum Number of Cycles	1e+07
Maximum Energy Error	0,1
Reference Energy Cycle	0
Initial Time Step	Program ...
Minimum Time Step	Program ...
Maximum Time Step	Program ...
Time Step Safety Factor	0,9
Characteristic Dimension	Diagonals
Automatic Mass Scaling	No

Solver Controls

Solve Units	mm, mg, ms
Beam Solution Type	Bending
Beam Time Step Safety Factor	0,5
Hex Integration Type	Exact
Shell Sublayers	3
Shell Shear Correction Factor	0,8333
Shell BWC Warp Correction	Yes
Shell Thickness Update	Nodal
Tet Integration	Average ...
Shell Inertia Update	Recompute
Density Update	Program ...
Minimum Velocity	1, e-003 m...
Maximum Velocity	1, e+013 ...
Radius Cutoff	1, e-003

B: Explicit Dynamics
Analysis Settings
Time: 2, e-002 s
30.04.2021 15:02

ANSYS 2020 R1 ACADEMIC

0,02 s

0,00 200,00 400,00 (mm)

Steps	End Time [s]
1	2, e-002
*	

1

1 Message No Selection Metric (mm, t, N, s, mV, mA) Degrees rad/s Celsius

Fixed support

The screenshot displays the ANSYS Academic Research Mechanical and CFD software interface. The main window shows a 3D model of a tapered shaft with a fixed support applied to its left end. The software is running an explicit dynamics analysis.

Context: Multiple Systems - Mechanical [ANSYS Academic Research Mechanical and CFD - ANSYS Academic Research Mechanical and CFD]

Environment: Display Selection Automation

Quick Launch: [Search bar]

Tools: Duplicate, Outline, Solve, Solver, Analysis, Named Selection, Coordinate System, Remote Point, Commands, Comment, Chart, Images, Section Plane, Annotation, Inertial, Loads, Force, Moment, Pressure, Supports, Frictionless, Displacement, Conditions, Direct FE, Tools, Views.

Outline:

- Project*
- Model (A4, B4)
 - Geometry
 - SYS\Solid
 - Materials
 - Pine wood
 - Coordinate Systems
 - Mesh
 - Static Structural (A5)
 - Analysis Settings
 - Fixed Support
 - Pressure
 - Solution (A6)
 - Solution Information
 - Total Deformation
 - Equivalent Stress
 - Explicit Dynamics (B5)
 - Initial Conditions
 - Analysis Settings
 - Fixed Support
 - Solution (B6)
 - Solution Information
 - Total Deformation
 - Equivalent Stress

Details of "Fixed Support"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Fixed Support
Suppressed	No

B: Explicit Dynamics
Fixed Support
Time: 2,e-002 s
30.04.2021 15:04

ANSYS 2020 R1 ACADEMIC

Fixed Support

0,00 50,00 100,00 (mm)

Graph: 2,e-2

Messages | Graph

1 Message No Selection Metric (mm, t, N, s, mV, mA) Degrees rad/s Celsius

Nastavení Total deformation a Equivalent Stress

The screenshot displays the ANSYS 2020 R1 Academic Mechanical interface. The main window shows a 3D model of a long, thin rectangular bar. The software is in the 'Solution' phase, and the 'Explicit Dynamics (B5)' solution is active. The 'Solution (B6)' details panel shows 'Status: Solve Required' and 'Post Processing: Beam Section Results: No'. The 'Equivalent Stress' result is visible in the 'Details of "Solution (B6)"' panel.

The 'Outline' pane on the left shows the project structure:

- Project*
- Model (A4, B4)
 - Geometry
 - SYS\Solid
 - Materials
 - Pine wood
 - Coordinate Systems
 - Mesh
 - Static Structural (A5)
 - Analysis Settings
 - Fixed Support
 - Pressure
 - Solution (A6)
 - Solution Information
 - Total Deformation
 - Equivalent Stress
 - Explicit Dynamics (B5)
 - Initial Conditions
 - Analysis Settings
 - Fixed Support
 - Solution (B6)
 - Solution Information
 - Total Deformation
 - Equivalent Stress

A red callout box labeled 'Definice' points to the 'Equivalent Stress' result in the 'Solution (B6)' sub-tree.

The 'Graph' pane at the bottom shows a plot of 'Time [s]' from 0 to 5, e-2. The 'Tabular Data' pane shows the following data:

Time [s]
1 1,1755e-038
2 5,9519e-003
3 1,1901e-002
4 1,7852e-002
5 2,0002e-002
6 2,1001e-002
7 2,2003e-002
8 2,3002e-002

The status bar at the bottom indicates '(43%) Solving the mathematical model...' and '1 Message No Selection Metric (mm, t, N, s, mV, mA) Degrees rad/s Celsius'.