ENGINEERING TOMORROW

POWER TO CHANGE
THROUG
DIGITALIZATION

WWW.INAS.RO

INAS
power to change

Ansys Mechanical 2024 R1 Course:

Ansys Geometry:

- **Day 1:**

- Ansys Geometry overview: General description of the Ansys Geometry interfaces (Design Modeler / SpaceClaim Direct Modeler), Sketching and 3D modeling techniques, Geometry simplification and repair, Working with design parameters

- **Workshop 1.1:** Using sketches as base objects for 3D operations
- **Workshop 1.2:** Slice operation, pattern operation, mid-surface creation, analysis tools
- **Workshop 1.3:** Finding geometry imperfections using various tools and simplifying / repairing of an assembly
- **Workshop 1.4:** Generation of a parameterized 3D model

Ansys Meshing:

- **Day 2:**

- Ansys Meshing overview: Finite Element Method overview, Generating a mesh, Creating named selections, Global mesh controls

- **Workshop 2.1:** Mesh generation using different methods
- **Workshop 2.2:** Using global controls for mesh sizing and refinement

- **Day 3:**

- Meshing methods (Tetrahedral, Hex dominant, Sweep, Multizone, Cartesian), Local mesh controls, Checking mesh quality

- **Workshop 3.1:** Mesh generation using different methods and their comparison
- **Workshop 3.2:** Demonstrate how multiple methods and local controls can be combined to create a conformal hybrid mesh on a multibody part

Ansys Mechanical:

- **Day 4:**

- Ansys Mechanical overview: General description of Mechanical 2024 R1 interface, Basics of static structural analysis, Mechanical application wizard, Pre & Postprocessing, Material properties, Boundary conditions and loads

- **Workshop 4.1:** Set up and solve a structural model for stress, deflection and safety factor
- **Workshop 4.2:** Static structural analysis of a small assembly

INAS
power to change

Ansys Mechanical 2024 R1 Course:

Ansys Mechanical:

- **Day 5:**

- Graphics control and selection, Outline tree and details, Connections, Coordinate systems, Object Generator

- **Workshop 5.1:** Connections definition: Contact controls. Contact results. Spot welds. Springs and beams
- **Workshop 5.2:** Joint configuration. Kinematic study of an assembly with rigid and flexible components

- **Day 6:**

- Multistep setup & controls, Defining and using parameters in Ansys Workbench, Updating CAD parameters, Parameter management

- **Workshop 6.1:** Multistep analysis of a small assembly
- **Workshop 6.2:** Multiple analysis scenarios using parameters

Ansys Mechanical:

- **Day 7:**

- Vibration basics, Modal analysis, Vibration with prestress, Solution setup, Modal results

- **Workshop 7.1:** Free vibration analysis of a component
- **Workshop 7.2:** Pre-stress modal analysis of the same model
- **Workshop 7.3:** Random vibration analysis

- **Day 8:**

- General tools and procedures useful in nonlinear structural analyses

- **Workshop 8.1:** Frictional contact simulation
- **Workshop 8.2:** Using interface treatment tools to simulate an interference fit between surfaces

Ansys Mechanical:

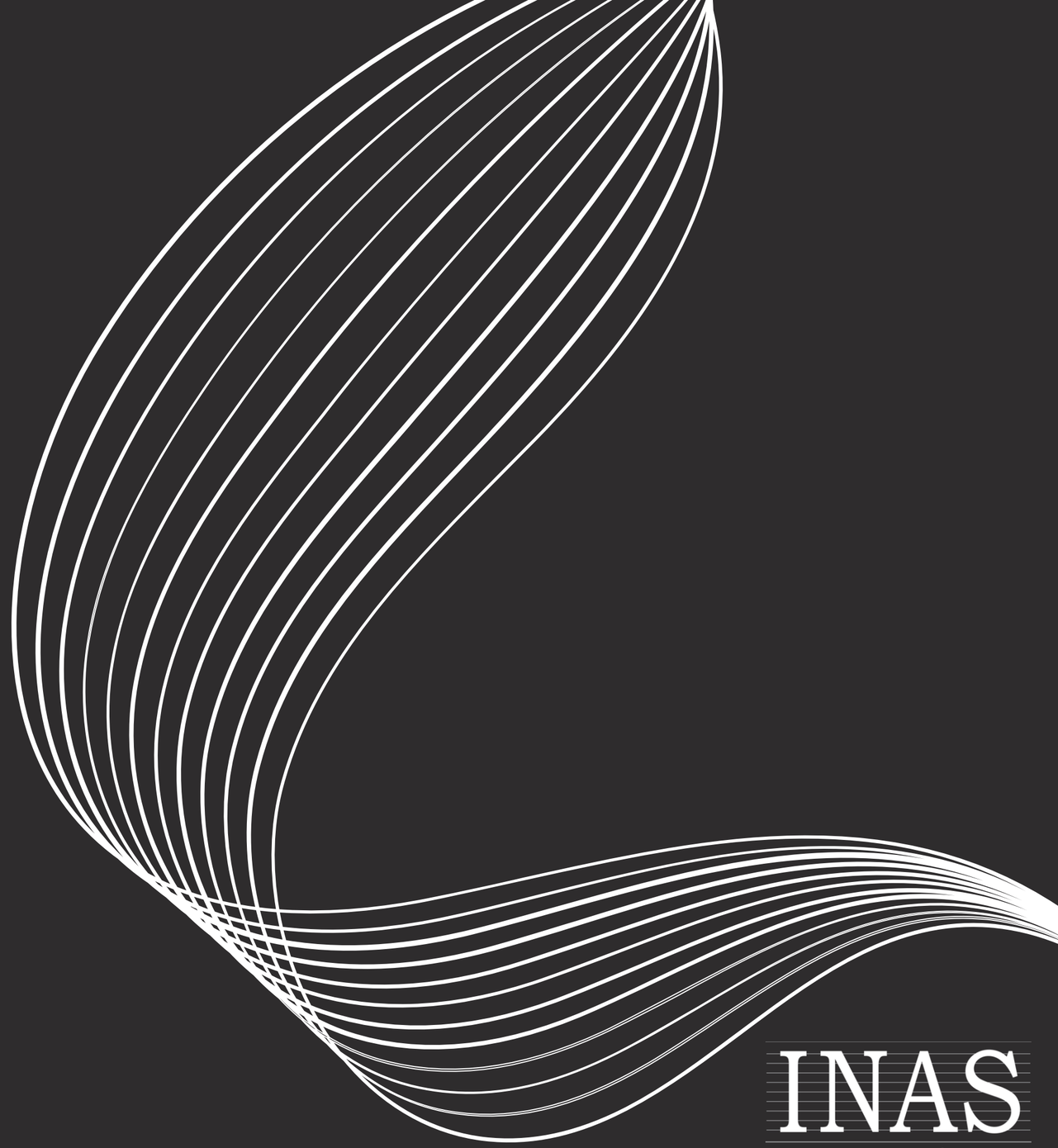
- **Day 9:**

- Explicit Dynamics Overview, Explicit dynamics simulation setup & controls, Impact analysis, Adaptive meshing

- **Workshop 9.1:** Circuit board drop test
- **Workshop 9.2:** Impact simulation using a SPH model
- **Workshop 9.3:** Simulation of the deep drawing process for a metal sheet (LS-Dyna)

INAS
power to change

THANK YOU!



INAS

power to change