

*Presentation to the report №1004  
«Simulation of Drawing-Forming by  
Magnetic-Pulse Deformation»*

Presented By

**Professor, Sc.D.** Kurlaev Nikolay Vasilievich, NSTU

**PhD.** Ryngach Nikolay Anatolevich, NSTU

**Aspirant Eng.** Ahmed Soliman M. Sherif, NSTU

# Introduction

- ❖ One of the methods of high-speed plastic deformation of materials is electromagnetic pulse exposure
- ❖ To study the process of forming thin-sheet parts of aircraft and other machines, numerical modeling is used based on models of deformable solid mechanics
- ❖ To simulate the electromagnetic pulsed exposure, it is necessary to include the Maxwell equations in the eddy current approximation in the model
- ❖ This method is based on high intensity pulsed magnetic fields as a load source, which cause the action of electrodynamic forces on a conductive material placed in a magnetic field. Under the influence of these forces, the workpiece accelerates to high speeds (from 100 to 450 m/s)

# Aims

- ✓ One of the main applications of the electromagnetic module of LS-DYNA program is magnetic pulse metals treatment by pressure
- ✓ Electromagnetic pulse interaction is a method of plastic forming of materials with high-speed deformations
- ✓ Electromagnetic fields are solved by the finite element method (FEM) for conductors coupled with the Boundary Element Method (BEM) for ambient air / insulators
- ✓ This paper considers theoretical principles and discusses the physical phenomena that occur during the formation of a sheet blank by an electromagnetic pulse
- ✓ An example of numerical simulation of electromagnetic forming in the electromagnetic analysis module with the resulting mesh shape of the workpiece without a corrugation defect is given

# Experimental model by LS-DYNA

4

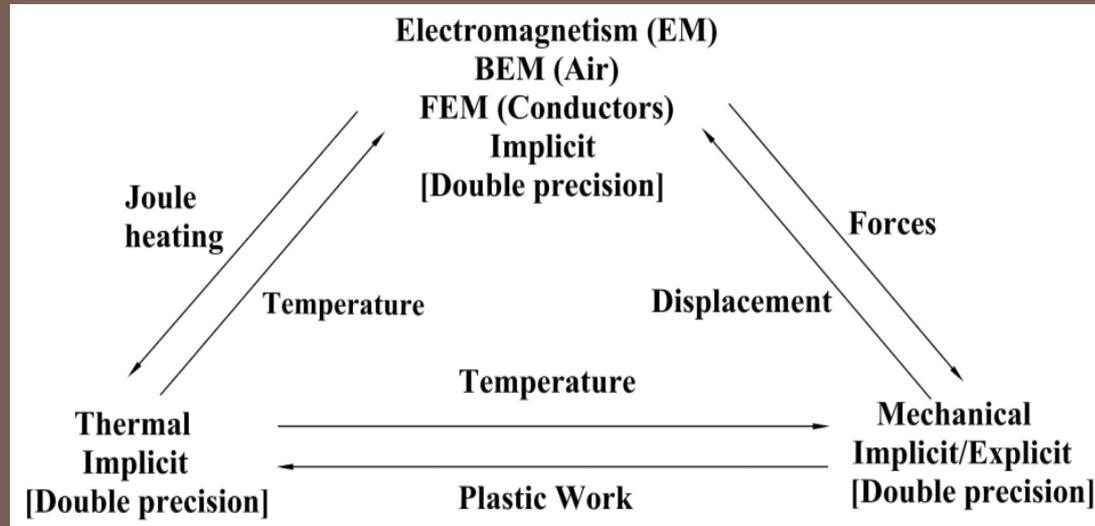


Fig. 1. Interactions between various LS-DYNA solvers.

Modeling in the EM module LS-DYNA program includes 3 stages:

- Creation of a finite element model and determination of all initial parameters
- Simulation of the created k file in the LS-DYNA program
- Analysis of the results

# Materials and Methods

## 1. Material Description

- ✓ The material model defines a mathematical function that describes the effect of certain physical parameters, i.e. strain, strain rate, and temperature, on the magnitude of stress
- ✓ Yield stress is the stress at which a material begins to deform plastically
- ✓ To describe the deformation of metallic materials, LS-DYNA provides a variety of classical yield criteria (von Mises, Hill, Tresca) and criteria that depend on the temperature and strain rate. Typically, LS-DYNA allows you to describe the material model for metallic and non-metallic materials
- ✓ The model geometry can only be discretized when using solid elements with insufficiently integrated and fully integrated formulations

## 2. Mechanical solver

- ✓ Once the electromagnetic fields have been computed, the Lorentz force is evaluated at the nodes and added to the mechanical solver
- ✓ Both mechanical and electromagnetic solvers have their own timestep, the mechanical timestep is smaller than the electromagnetic in 10 times

## 3. Thermal solver

- ✓ Thermal solver allowing to update the temperature
- ✓ LS-Dyna provides isotropic, orthotropic and anisotropic thermal models with phase transition thermal models. Based on the Burgess model, temperature is used in the electromagnetic equation of state to update the conductivity. Joule heating and plastic dissipation heating are provided by a thermal solver

# Creating a finite element model

7

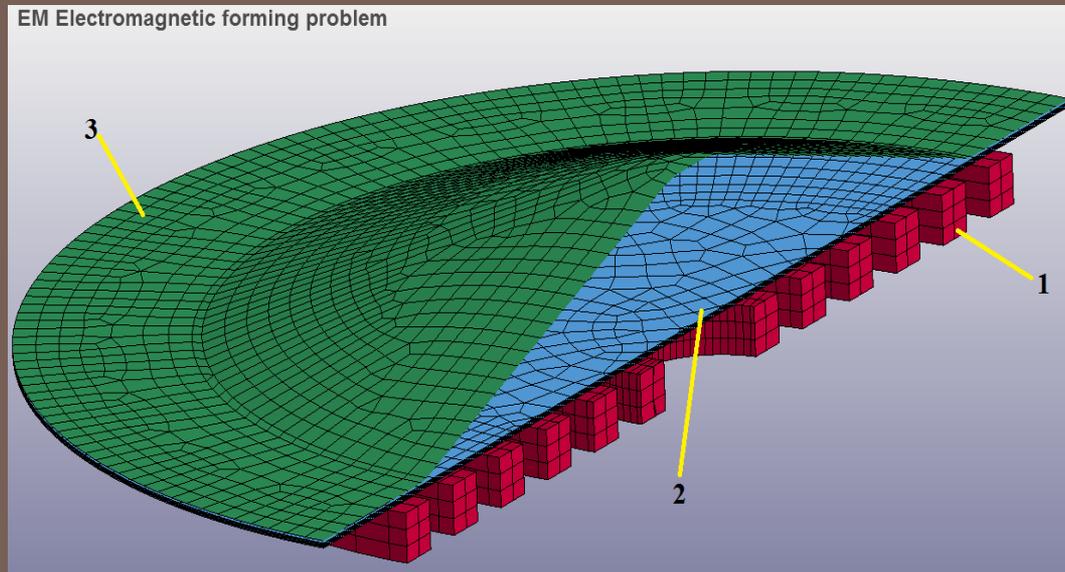


Figure 2. Longitudinal section of finite element modeling with six turns of the coil: 1 – coil; 2 – blank; 3 – die.

Figure 2 shows a longitudinal section of the grid model at the initial time. For the coil and the workpiece, a hexagonal grid (solid) was built, and for the die, (shell) elements were used

# Results

8

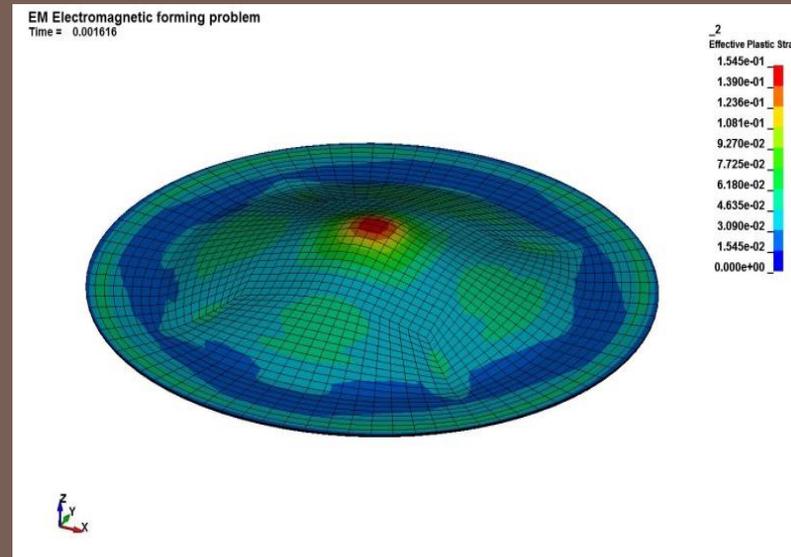


Figure 3. Final shape, plastic deformation on the workpiece.

- 1) LS-Dyna program was chosen for modeling the processes of magnetic pulse deformation, which allows us to solve these problems with great accuracy, and use alternately the following modules in the calculation process: mechanical, thermal, and electromagnetism
- 2) Figure 3 shows the shape of the workpiece during forming with a clamp

- 3) An additional energy was required for applying the clamp
- 4) After a numerical calculation in the LS-DYNA program, we constructed a nomogram (fig.4) to determine the maximum pressure of the magnetic field (MF) on the workpiece depending on the power consumption of the generator, the inductor-workpiece system inductance, the number of turns of the inductor and the current strip width
- 5) The horizontal part is 1.5 mm, i.e. corresponds to the original thickness of the workpiece. The clamp does not allow increasing the thickness of the workpiece in this zone. And the thickness in the central part is reduced to 1.23 mm.

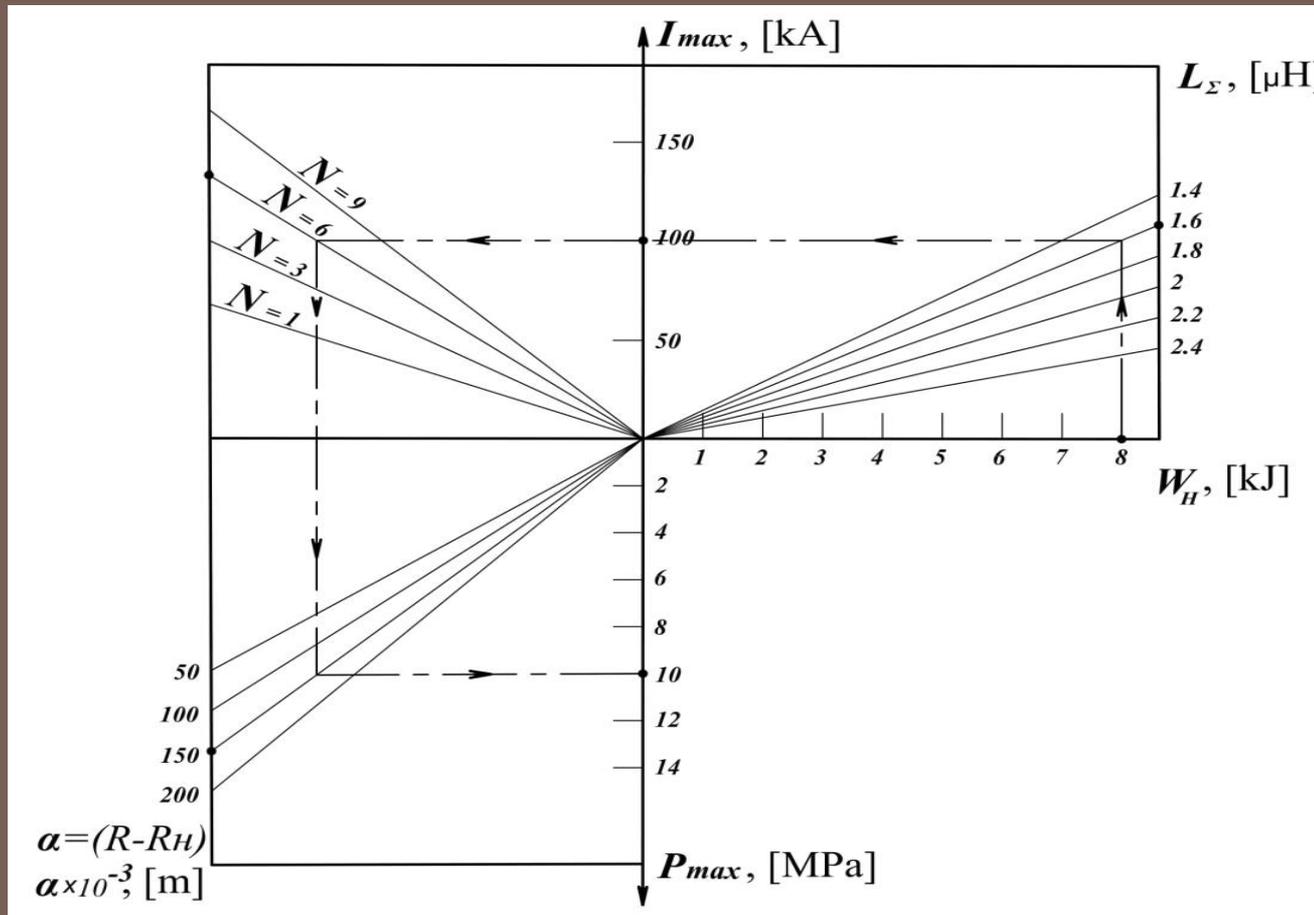


Figure 4. Nomogram for determining the amplitude of the maximum pressure of (PMF) from energy for drawing-forming at various parameters of the inductor systems.

Thanks for your attention!