



**AGH**

AKADEMIA GÓRNICZO-HUTNICZA  
IM. STANISŁAWA STASZICA W KRAKOWIE

# ***ABAQUS***

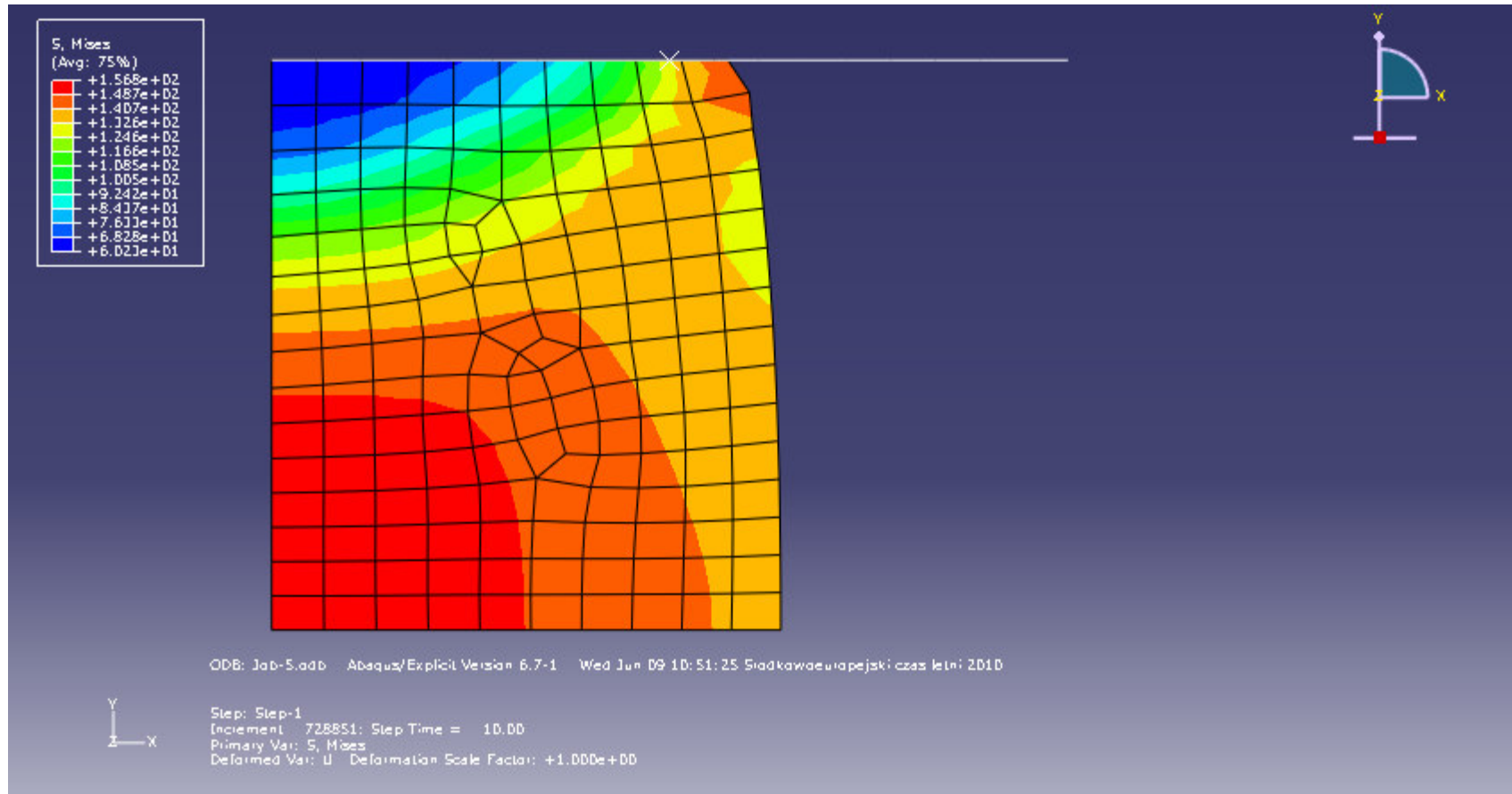
## ***Exercise 3***

### ***Plastic deformation***

**dr inż. Piotr Kustra**



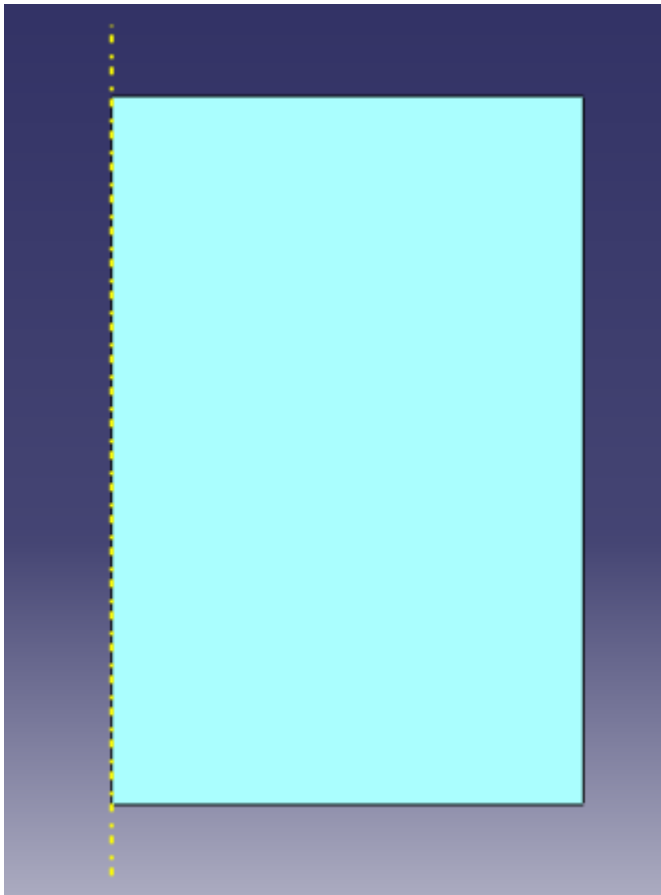
# Plastic deformation in axisymmetric model



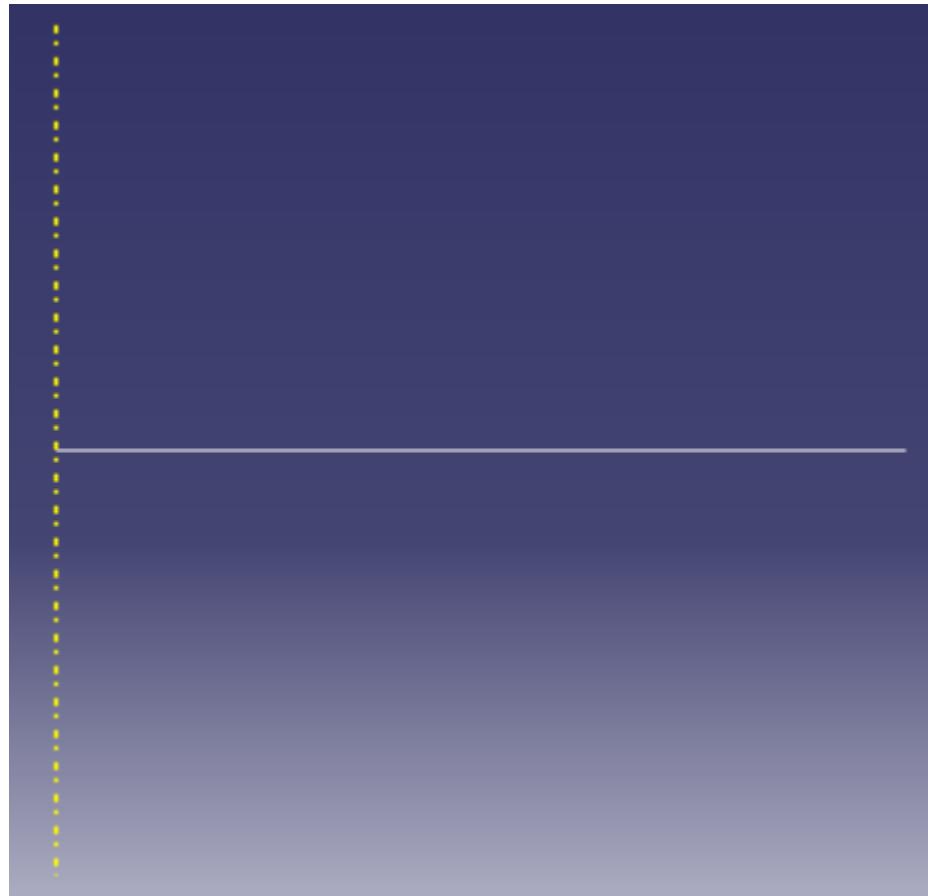


## Sample and die - axisymmetric

shell



wire

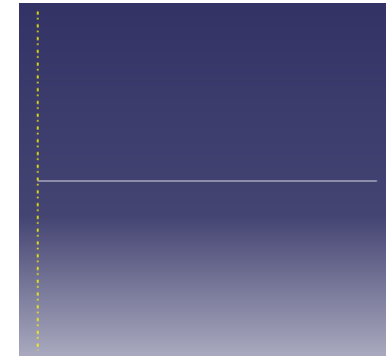
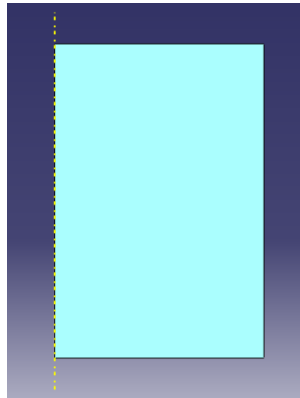




# Sample and die - axisymmetric

shell

wire



Create Part

Name: Part-3

Modeling Space

3D  2D Planar  Axisymmetric

Type

Deformable  
 Discrete rigid  
 Analytical rigid

Options

Include twist

Base Feature

Shell  
 Wire  
 Point

Approximate size: 200

Continue... Cancel

Create Part

Name: Part-3

Modeling Space

3D  2D Planar  Axisymmetric

Type

Deformable  
 Discrete rigid  
 Analytical rigid

Options

None available

Base Feature

Wire  
 Point

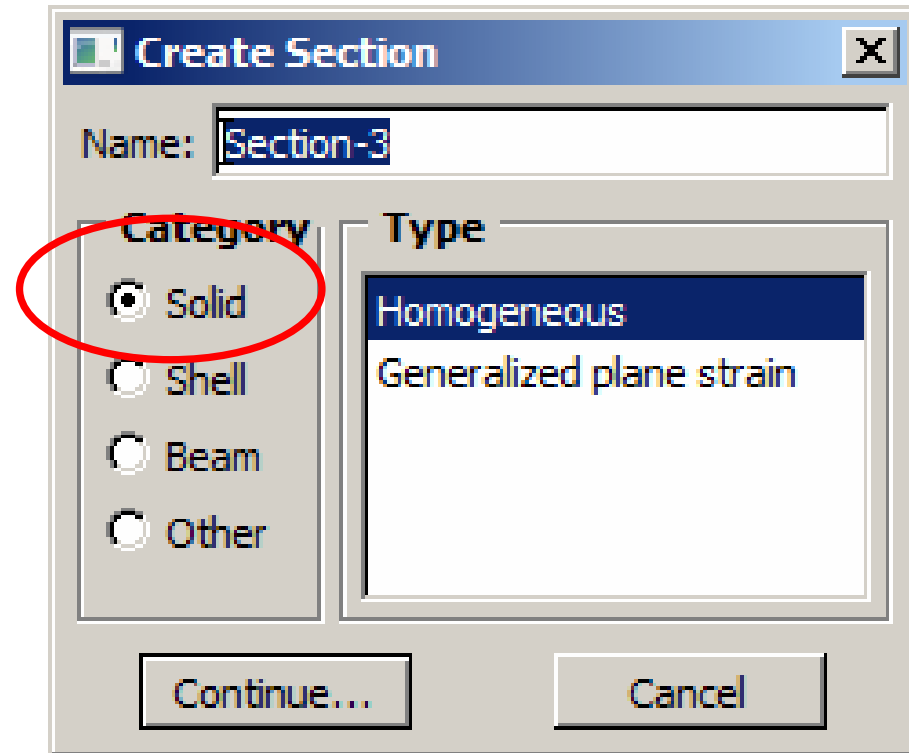
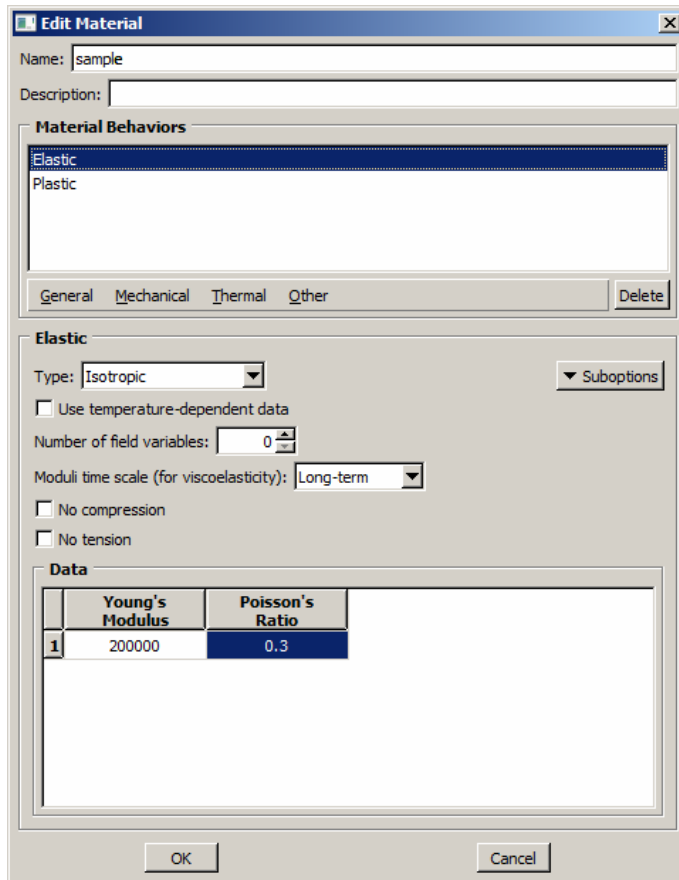
Approximate size: 200

Continue... Cancel



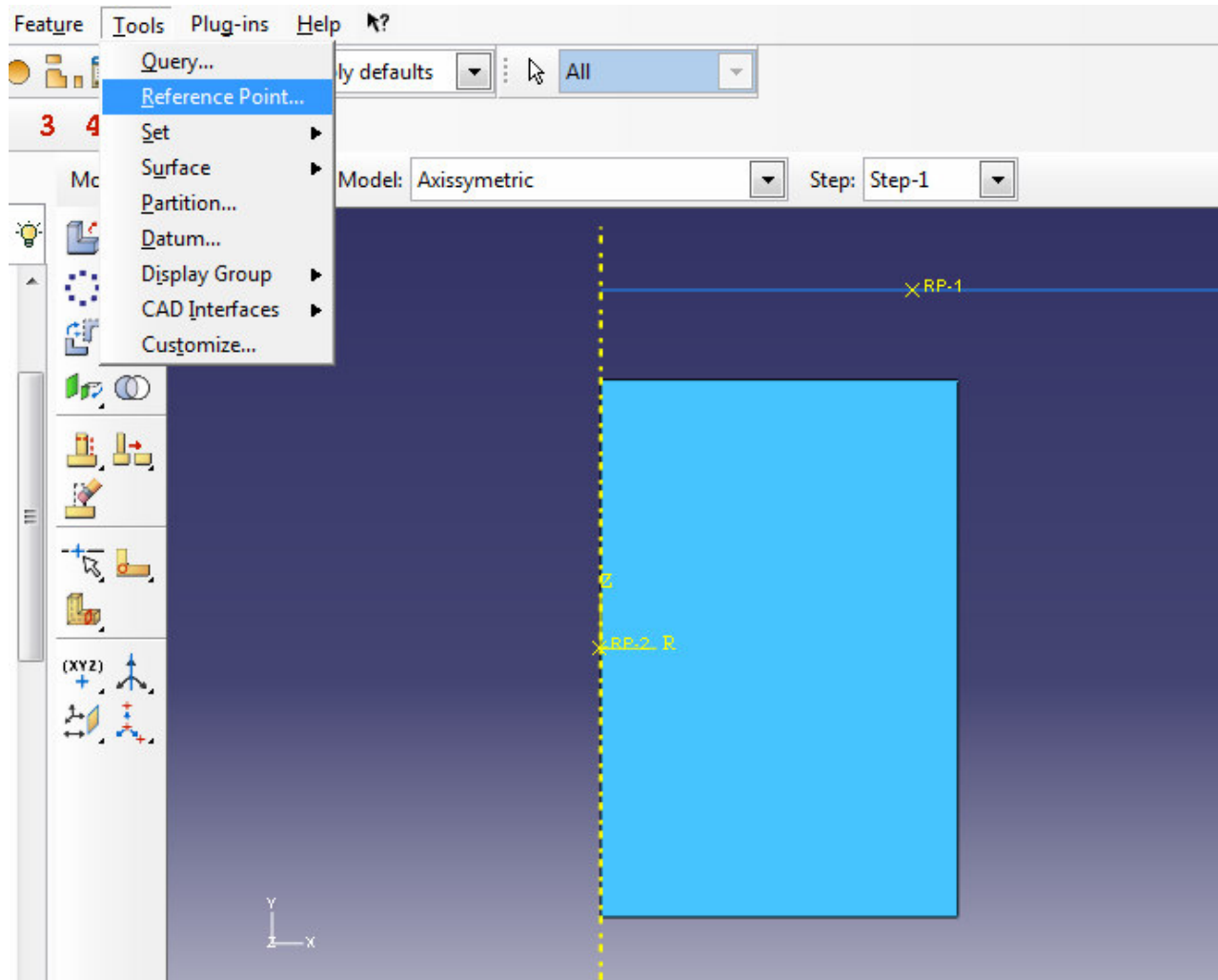
# Materials and sections

## Elasto-plastic material for sample





# Assembly - Reference Point





# Analyze type

**Create Step**

Name:

Insert new step after

- Initial
- Step-1**

Procedure type:

- Coupled temp-displacement
- Dynamic, Implicit
- Geostatic
- Soils
- Static, General**
- Static, Riks
- Visco

Name: Step-1

Type: Static, General

Basic | Incrementation | Other

Description:

Time period:

Nlgeom: On

Automatic stabilization:

**Edit Step**

Name: Step-1

Type: Static, General

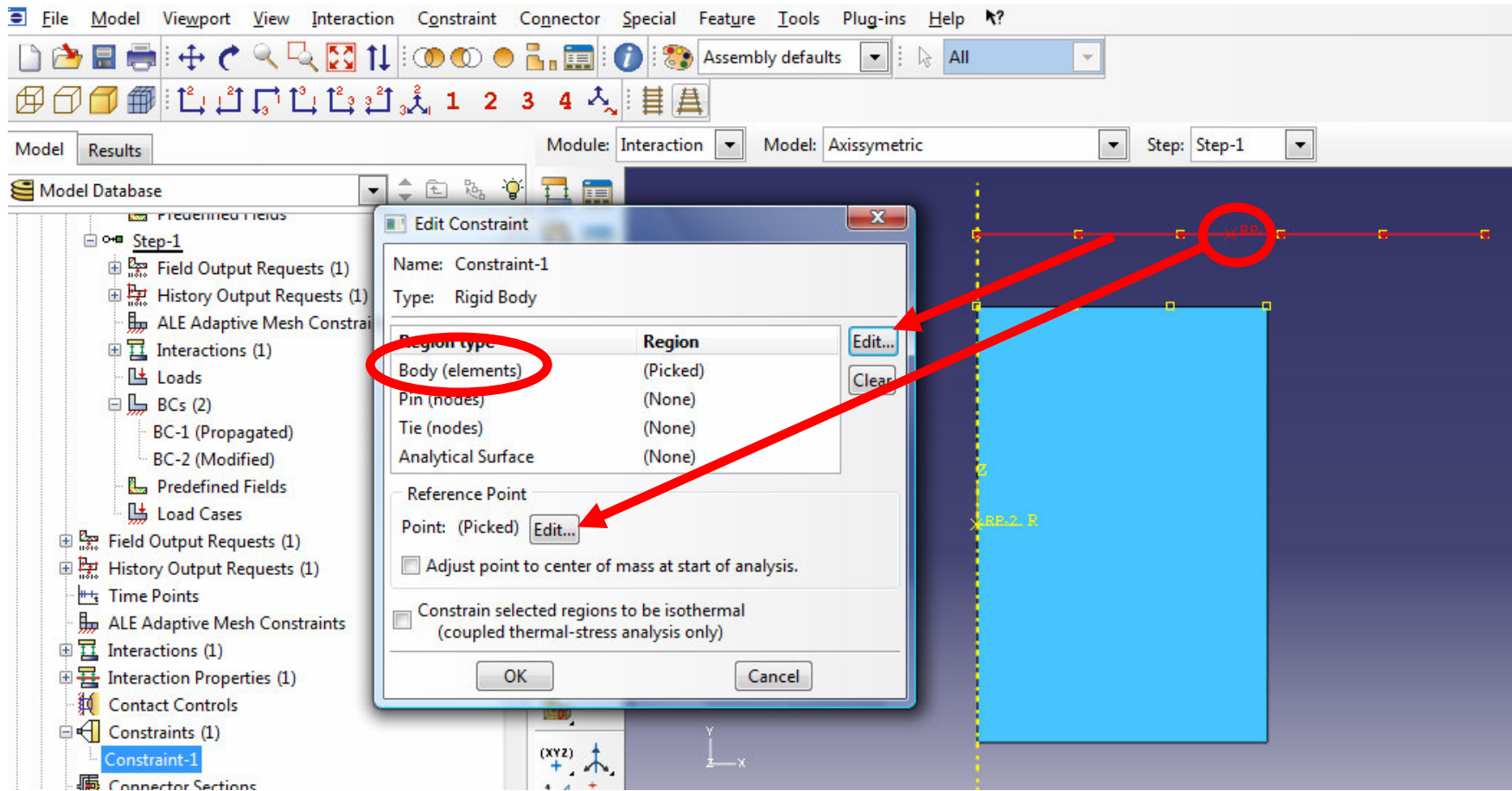
Basic | Incrementation | Other

Type:  Automatic  Fixed

Maximum number of increments:

Increment size:	Initial	Minimum	Maximum
	<input type="text" value="1"/>	<input type="text" value="0.0001"/>	<input type="text" value="1"/>

# Wiązanie RF z częścią



File Model Viewport View Interaction Constraint Connector Special Feature Tools Plug-ins Help

Assembly defaults All

Model Results Module: Interaction Model: Axisymmetric Step: Step-1

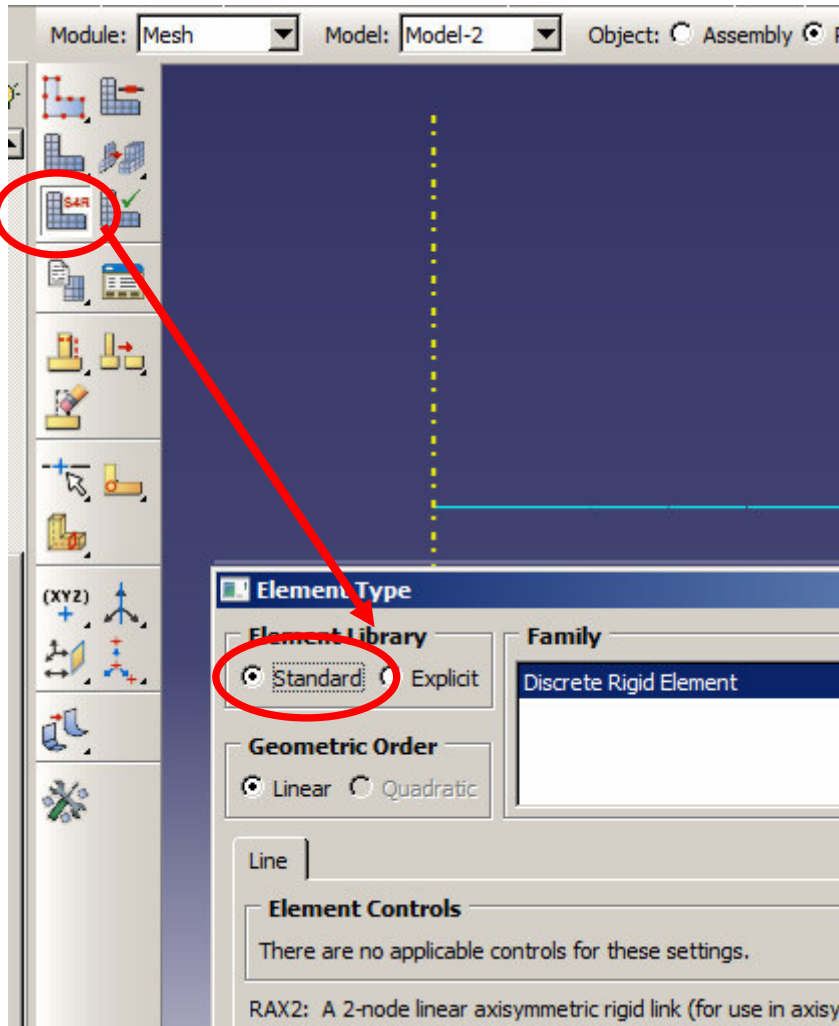
Model Database

- Step-1
  - Field Output Requests (1)
  - History Output Requests (1)
  - ALE Adaptive Mesh Constraints (1)
  - Interactions (1)
  - Loads
  - BCs (2)
    - BC-1 (Propagated)
    - BC-2 (Modified)
  - Predefined Fields
  - Load Cases
  - Field Output Requests (1)
  - History Output Requests (1)
  - Time Points
  - ALE Adaptive Mesh Constraints
  - Interactions (1)
  - Interaction Properties (1)
  - Contact Controls
  - Constraints (1)
    - Constraint-1
  - Connector Sections



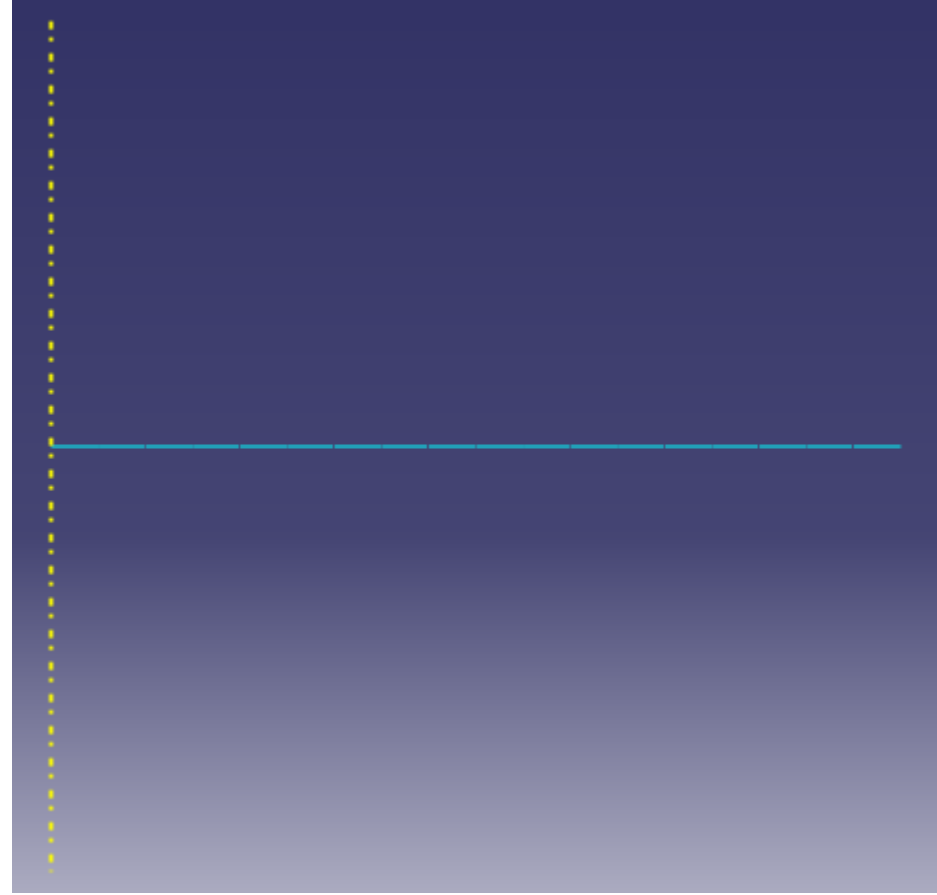
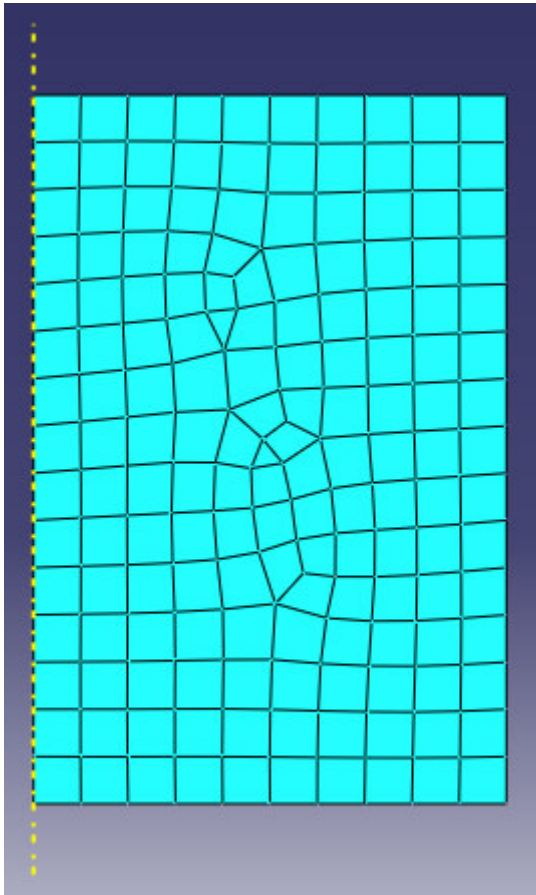


# MESH type for tool and sample



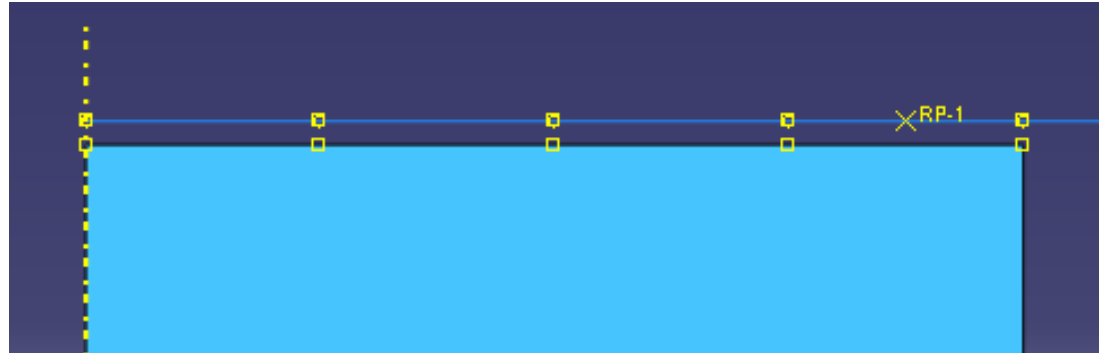
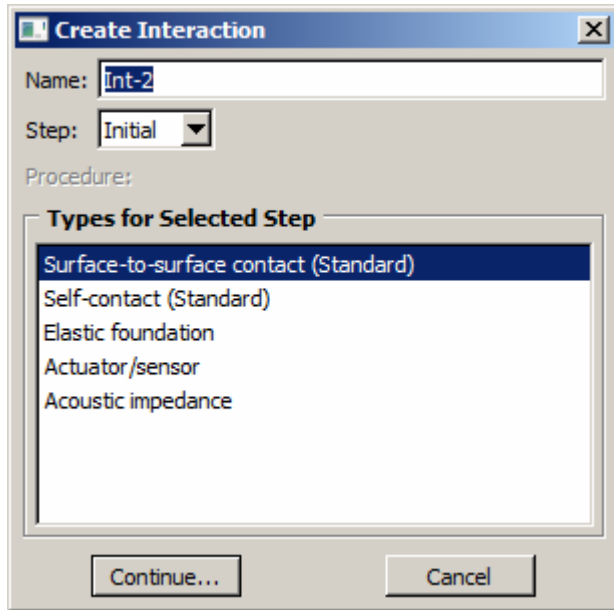


# Siatki MES



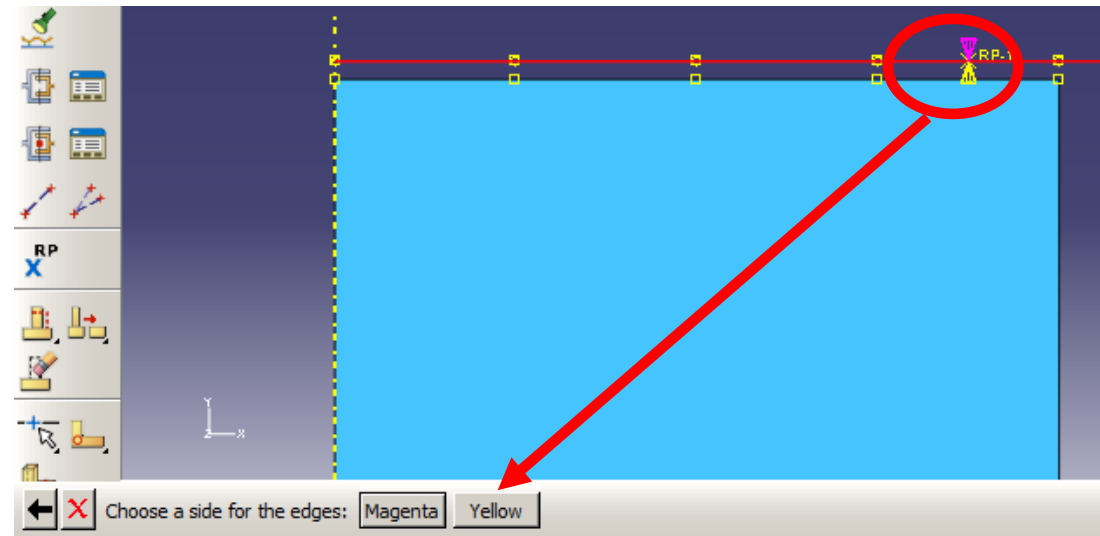


# Contact interaction



**Master surface – die!!!**

**Color side - Yellow**





# Contact definition

**Edit Interaction**

Name: Int-1  
Type: Surface-to-surface contact (Standard)  
Step: Initial

Master surface: (Picked)

Slave surface: (Picked)

Sliding formulation:  Finite sliding  Small sliding

Discretization method:

Exclude shell/membrane element thickness

Degree of smoothing for master surface:

Use supplementary contact points:  Selectively  Never  Always

Constraint position:  Node centered  Face centered

Contact tracking:  Single configuration (state)  Two configurations (path)

Slave Node/Surface Adjustment

No adjustment  
 Adjust only to remove overdosure  
 Specify tolerance for adjustment zone:   
 Adjust slave nodes in set:

Contact interaction property:

Options:

Contact controls:

**Edit Contact Property**

Name: IntProp-1

**Contact Property Options**

**Tangential Behavior**  
Normal Behavior

**Tangential Behavior**

Friction formulation:

Directionality:  Isotropic  Anisotropic (Standard only)

Use slip-rate-dependent data  
 Use contact-pressure-dependent data  
 Use temperature-dependent data

Number of field variables:

Friction Coeff
0.2



# Boundary condition

The screenshot displays the Abaqus software interface. On the left, the 'Model Tree' shows a hierarchy of elements: Material-1, Sections (1), Profiles, Assembly, Instances (2), Position Constraints, Features (3), Sets, Surfaces, Connector Assignments, Engineering Features, Steps (2), Initial, Interactions (1), BCs (1), BC-1 (Created), Predefined Fields, Step-1, Field Output Requests (1), History Output Requests, ALE Adaptive Mesh Cons, Interactions (1), Loads, BCs (2), Predefined Fields, and Load Cases. A red circle highlights 'BC-1 (Created)' in the BCs (1) folder, with a red arrow pointing to the 'Edit Boundary Condition' dialog box.

The main window shows a blue rectangular model with a vertical dashed yellow line labeled 'RP-1' representing a plane of symmetry. The bottom edge of the model is highlighted in red, indicating the selected region for the boundary condition. A coordinate system (XYZ) is visible at the bottom left.

The 'Edit Boundary Condition' dialog box is open, showing the following settings:

- Name: BC-1
- Type: Symmetry/Antisymmetry/Encastre (highlighted with a red circle)
- Step: Initial
- Region: (Picked) Edit Region...
- Options:
  - XSYMM ( $U_1 = UR_2 = UR_3 = 0$ )
  - YSYMM ( $U_2 = UR_1 = UR_3 = 0$ )
  - ZSYMM ( $U_3 = UR_1 = UR_2 = 0$ )
  - XASYMM ( $U_2 = U_3 = UR_1 = 0$ ; Abaqus/St
  - YASYMM ( $U_1 = U_3 = UR_2 = 0$ ; Abaqus/St
  - ZASYMM ( $U_1 = U_2 = UR_3 = 0$ ; Abaqus/St
  - PINNED ( $U_1 = U_2 = U_3 = 0$ )
  - ENCASTRE ( $U_1 = U_2 = U_3 = UR_1 = UR_2 =$

Buttons for 'OK' and 'Cancel' are visible at the bottom of the dialog box.



# Boundary condition

The screenshot shows the Abaqus software interface. On the left is the 'Model Tree' with 'BC-2 (Created)' selected. The main window displays the 'Edit Boundary Condition' dialog box for 'BC-2'. The dialog shows 'Type: Displacement/Rotation', 'Step: Step-1 (Static, General)', and 'Region: (Picked)'. The 'Amplitude' dropdown is set to 'Amp-1' and is circled in red. A red arrow points from this dropdown to the 'Edit Amplitude' dialog box. The 'Edit Amplitude' dialog shows 'Name: Amp-1', 'Type: Tabular', and 'Time span: Step time'. It has a table with two rows: (1, 0, 0) and (2, 10, 1). The second row is highlighted in blue. The 'Amplitude Data' tab is active.

**Region for displacement is reference point !!!!!!!**

	Time/Frequency	Amplitude
1	0	0
2	10	1



## Results – stress intensity [MPa]

