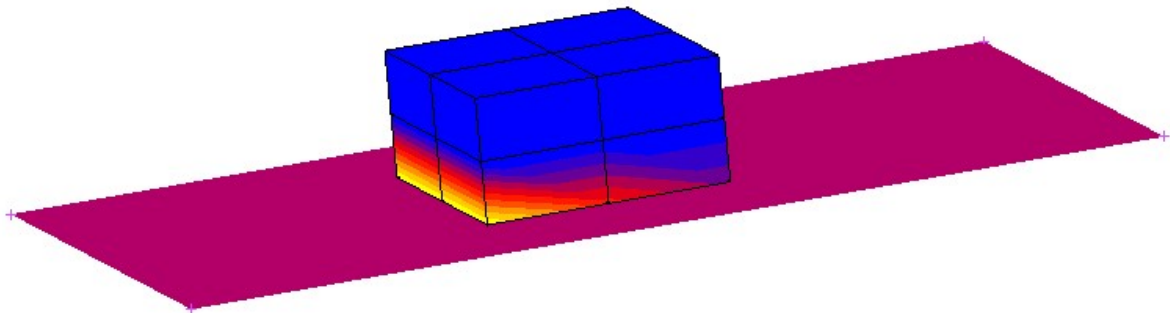# ThermoMechanical Test

1. **Files**
    1.1 New
    1.2 Ok
2. File: Save as: *Problem_Termo-mechaniczny* ("mud" extension is automatic)
3. On the **top Menu line**:
    verify Analysis dimension and type.
4. **Geometry & Mesh**
    4.1 Basic Manipulation: Length Unit: Inch
    4.2 Basic Manipulation: Geometry & Mesh
        a) Add Elements:
            type: node(-1,-1,0) Enter
                node(1,-1,0) Enter
                node(1,1,0) Enter
                node(-1,1,0) Enter
        b) Add Surfaces:
            type: point(1,-1,0) Enter
                point(-1,-1,0) Enter
                point(-1,1,0) Enter
                point(1,1,0) Enter
        c) Ok
    4.4 Operations: Move
        a) Scale Factors: 4 2 1
        b) Surfaces
            - All Existing
        c) Reset
        d) Translations: 1.8 0 0
        e) Surfaces
            - All Existing
        f) Ok
    4.5 Operations: Subdivide
        a) Elements
            - All Existing
        b) Ok
    4.5 Operations: Expand
        a) translations: 0 0 0.5
        b) repetitions: 2
        c) Expand Elements
            - All Existing
        d) Ok
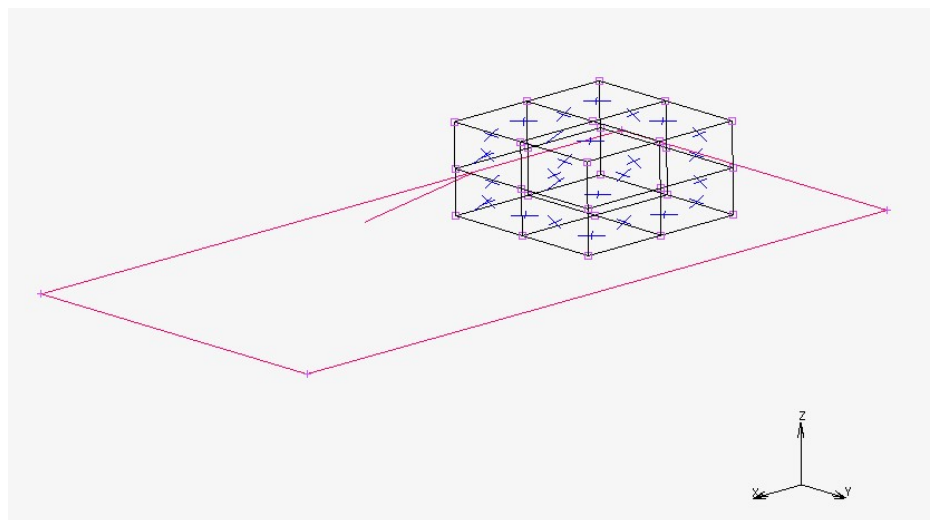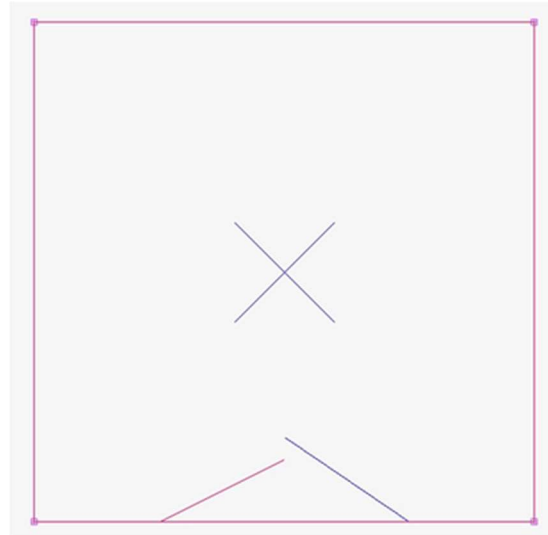   4.6 Operations: Sweep
        a) Remove Unused: nodes
        b) All
        c) Ok
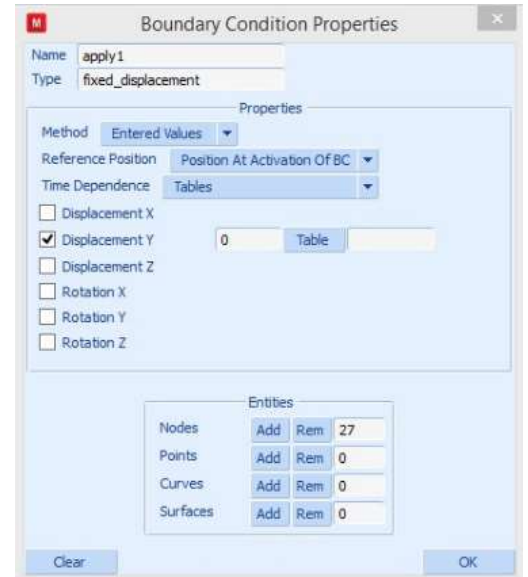   4.7 Basic Manipulation: Renumber
        a) All Geometry And Mesh
        b) Ok

5. Boundary conditions

5.1 New (Structural): Fixed Displacement
- a) Check Displacement Y = 0
- b) Nodes Add
  - All existing
- c) Ok

5.2 New (structural): Gravity load
- a) Acceleration Z = -9.81
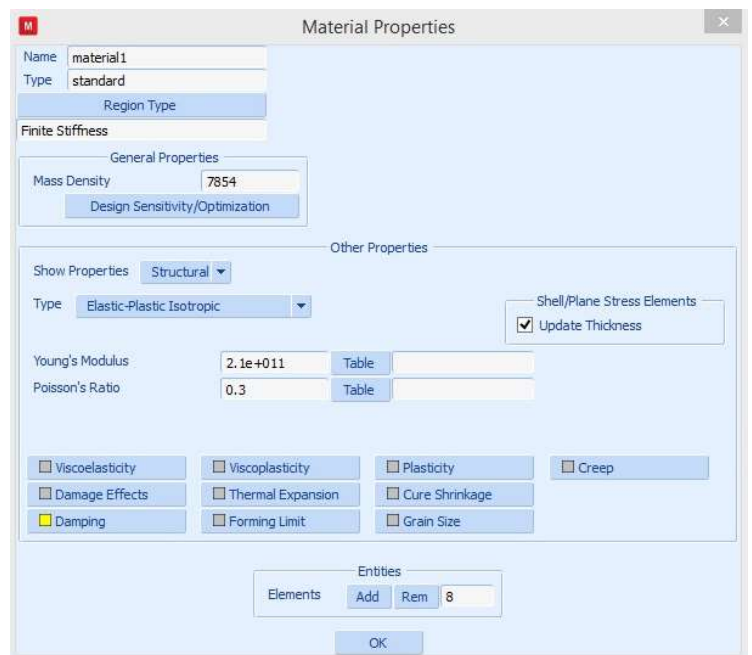- b) Elements Add
  - All Existing
- c) Ok

6. Initial Conditions

6.1 New (Thermal): Temperature
- a) Temperature = 0
- b) Select TOP and BOTTOM
- c) Nodes Add
  - All Existing
- d) Ok

6.2 New (structural): Velocity
- a) Velocity X: 4.905
- b) Nodes Add
  - All Existing
- c) Ok

7. Material Properties

7.1 New: Finite Stiffness region: Standard
- a) Mass Density = 7854
- b) Young's Modulus = 2.1e11
- c) Poisson's Ratio = 0.3
- d) Damping
  - Damping
  - Multiplier: 0.3
  - Ok
- e) Show properties: change to **Thermal**
- f) Conductivity K = 60.5
- g) Specific Heat = 434
- h) Mass density 7854
- i) Elements Add
  - All existing
- k) Ok

8. Contact
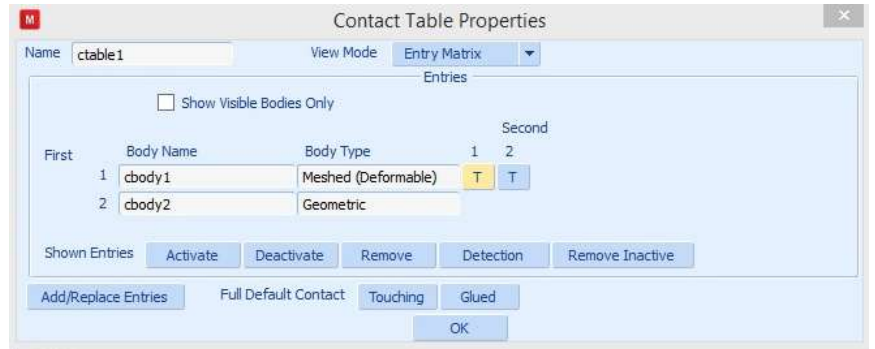
8.1 New: Meshed (Deformable)

    a) Elements Add

        - All Existing

    b) Ok

8.2 New: Geometric

    a) 3-D Surface Add

        - All Existing

    b) Ok

8.3 Contact Tables: New
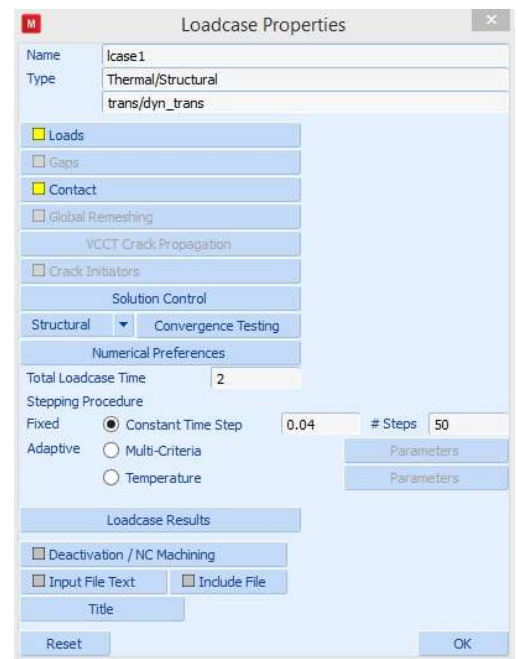
    a) Full Default Contact: Touching

    b) Entries 1

        - Contact Interaction: Edit

            - Friction

                -Friction coefficient = 0.5

                - Ok

            -Ok

        - Ok

    c)The same steps for Entries 2

    d) Ok

9. Loadcases

9.1 New: Transient/Dynamic transient

    a) Contact

        - Contact table

            - ctable1

            - Ok

        - Ok

    b) Solution Control

        - Non-Positive Definite

        - Ok

    c) Convergence Testing

        -Displacements

        - Ok

    d) Total Loadcase Time: 2

    e) # Steps = 50

    f) Ok

10. Jobs

10.1 New: *Thermal/Structural*

    a) choose **Large Strain**

    b) in Available Select: **lcase1**

        b) Analysis Options

            - Lumped Mass & Capacity

            - Ok

        c) Contact control

            - Friction Model: Arctangent(Coulomb)

- Relative Velocity Threshold = 0.1
- Advanced Contact Control
- Separation Force = 1e11
- Ok
- Ok

d) Job Results
- Select Equivalent Von Misses Stress
- Temperature
- Ok

e) Job parameters
- Conversion Factor 1e3
- Ok

f) Run
- Submit (1)
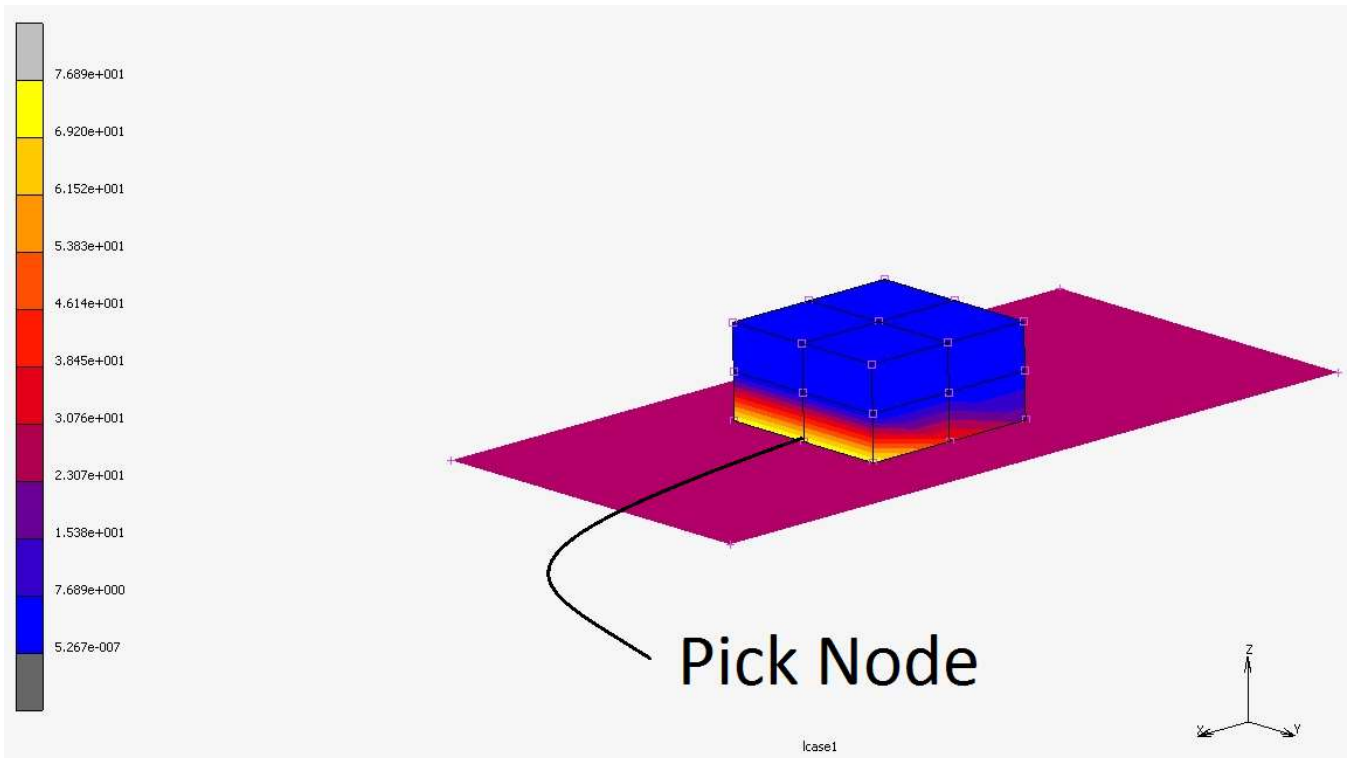- Monitor
*(After execution - **Open Post File**) or:*
g) Ok, Ok and File: (Result) Open Default

10.2 Save

11. Results
11.1 Model Plot Results

a) **Scalar Plot**: Style: Contour Bands
      b) Scalar: temperature
      c) Skip to 50 (i.e.: show the last increment) – or "*play the movie*"
d) Ok

11.2 **History Plot**

a) Set location (*select middle bottom node - **see above** and accept, e.g. **RMB***)
b) All Incs
c) Add Curves
      - on X axis: Data Carrier Type **Global**= Variable **Time**
      - on Y axis: Data Carrier Type Location (Node)
            - from the list below select node (here it was Node 22)
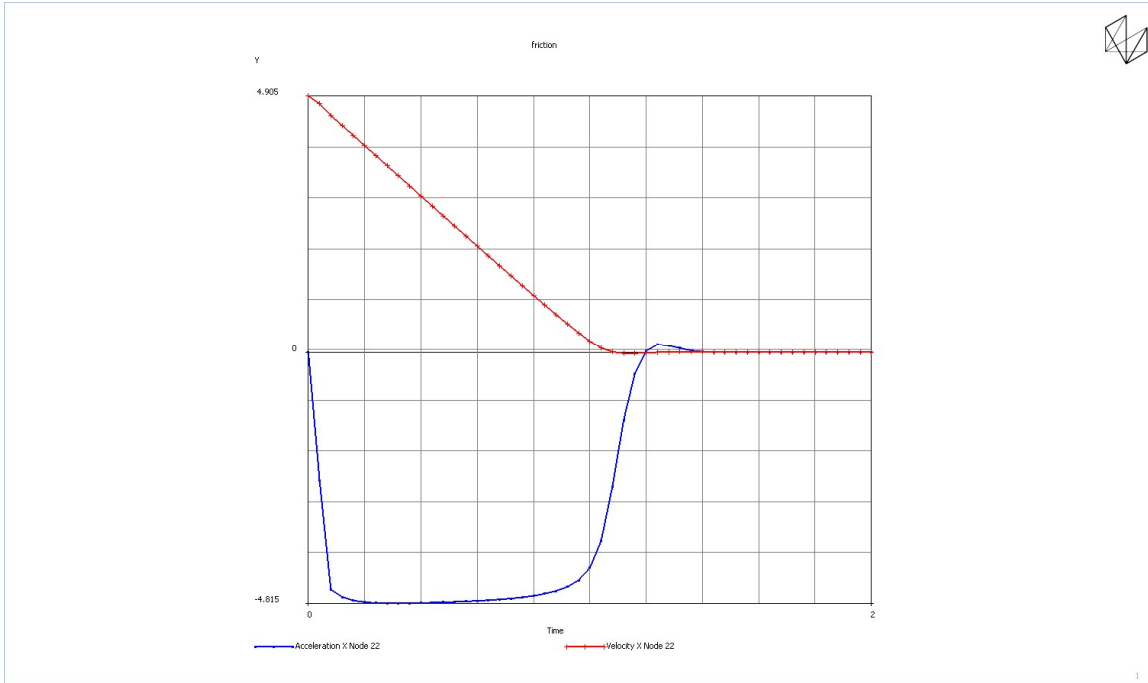   - Variable Velocity X
         Add Curve, Fit

Next curve, like above,
> On X axis: Time
> On Y axis: Acceleration X
> **Add Curve, Fit**

New curve, like above (but clear curves, as the Y -scale is different)
> On X axis: Time
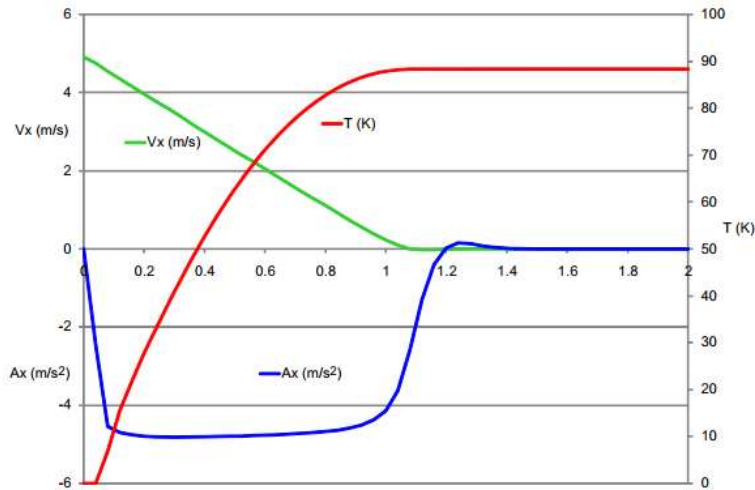> On Y axis: Temperature (Integration point)
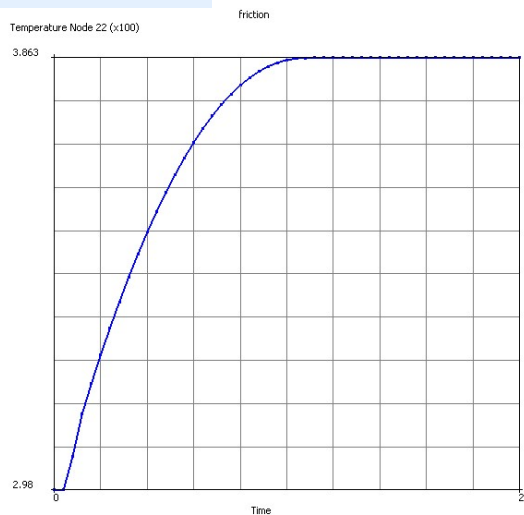
Figure 4.2-11   Velocity, Acceleration and Temperature History of Leading Node

Notice that the effect of friction was not 100% since the block should come to a stop at 1 sec. This was due to the ever slipping friction model. Rigid body dynamics gives:

$$\ddot{u} = -\mu g \quad ; \quad \dot{u} = -\mu g t + \dot{u}_0 \quad ; \quad u = -\mu g \frac{t^2}{2} + \dot{u}_0 t + u_0$$

where the initial velocity was selected as $\dot{u}_0 = \mu g t_s$. Where t_s is the stopping time or 1 second.

Also from the friction heating, the friction force moves through a distance and this mechanical energy is converted to thermal energy. This thermal energy is input to the heat transfer portion of the solution. Equating the conversion factor times the kinetic energy and accounting that for rigid contact only half of the frictional heating is added to the block, the average rise in temperature for a block that comes to rest from an initial velocity of $\dot{u}_0$, becomes:

$$\Delta T = \frac{1}{4} conv_{factor} \left(\frac{\dot{u}_0^2}{c_p} \right)$$

In this case, the rise in temperature is 13.86 K. Why is the block hotter at the leading bottom edge? What would you do to improve the results?

How does this compare with the Marc predictions? To answer this, we can close the post file and add another load case that is 1e6 seconds long allowing the block to come to thermal equilibrium, namely uniform temperature.