

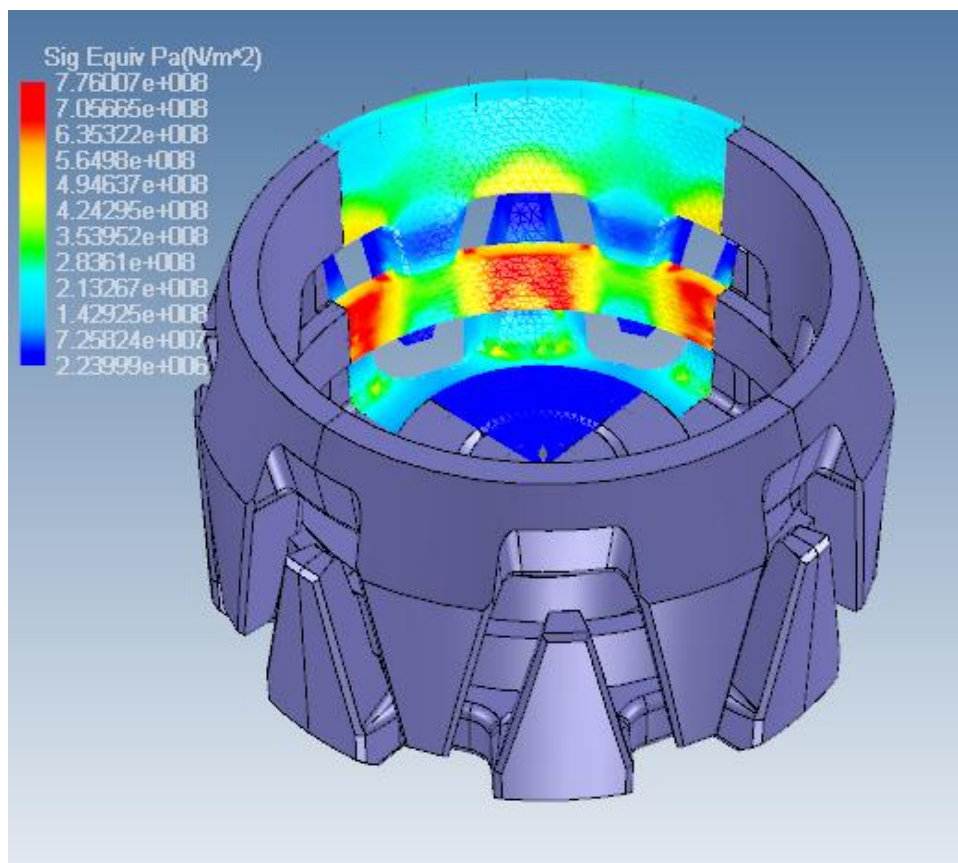
## Centrifuge Model

The centrifuge model demonstrates how to perform a rotational spinning centrifugal load analysis. It also demonstrates how to group different parts with/without the intended initial bonding for contact analysis, and to demonstrate the tolerance in solid model with initial touching contact.

Using only the quarter model with proper symmetry boundary condition, the result is the same as analyzing the whole model, but can be solved faster.

This model applies the automatic contact condition so the upper and the lower portion of the centrifuge will interlock together. In many legacy FEA tools, when contact analysis is involved, it requires the loading/constraint be applied in steps of very small increment. This model demonstrates the robustness of the automatic contact ability, and will automatically apply non-penetrating contact condition without any user intervention.

Suggested what-if analysis: change the contact friction; use half model rather than quarter model; change rotation speed.



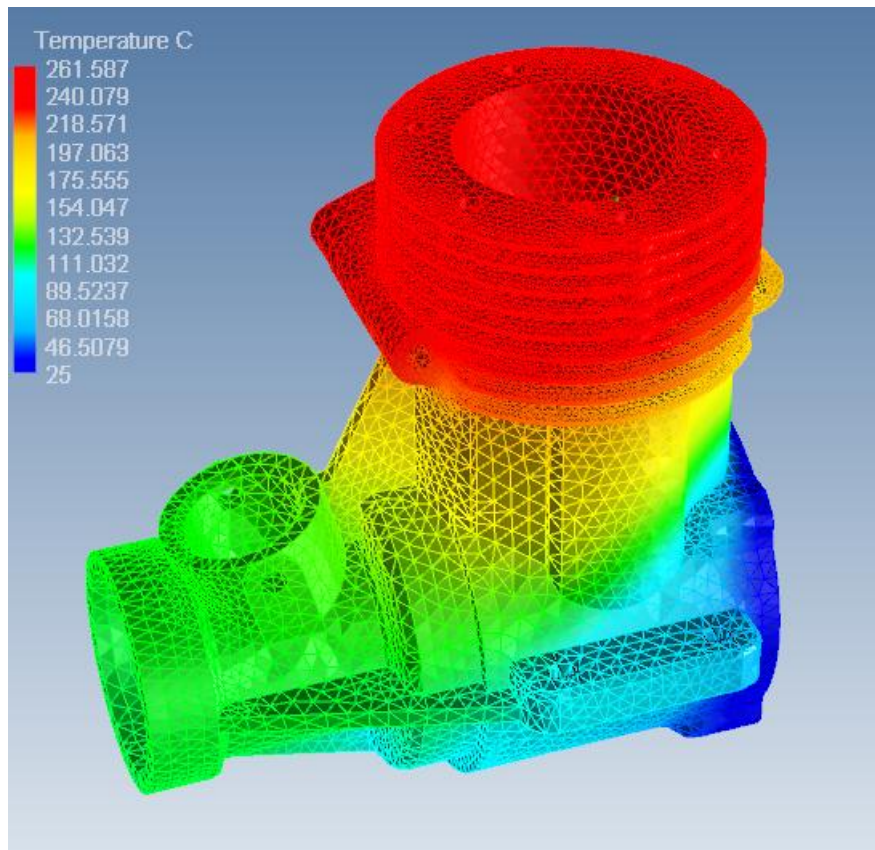
## Engine Block Model

The small 2-stroke hobby engine model demonstrates how to do a thermal-stress coupled analysis, and show how to set the proper thermal boundary condition to predict the thermal induced stress deformation.

The combustion heat generation effect is modeled using an equivalent heat flux in the interior of the cylinder wall area. It is analyzed by applying a known seating temperature at the rear end of the engine block. The only stress boundary condition is the fixed bolting constraint. It automatically computes both the temperature distribution and the induced deformation, mostly around the engine block.

In many legacy FEA tools, when thermal and stress coupled analysis is desired, usually a thermal heat transfer analysis has to be done first, and then overlaying the temperature result on the stress model to get the thermal deformation and stress. Not only such segregated method is not accurate, it may never converge or give accurate result if material properties have temperature dependency. IronCad Multiphsics simulates these types of coupled problem directly by solving the heat transfer and stress equilibrium together, and realistically simulate the physics as it happens.

Suggested what-if analysis: change the inner combustion heat flux intensity; apply surface convective heat loss on the engine block fins to see how it can decrease the steady-state temperature.



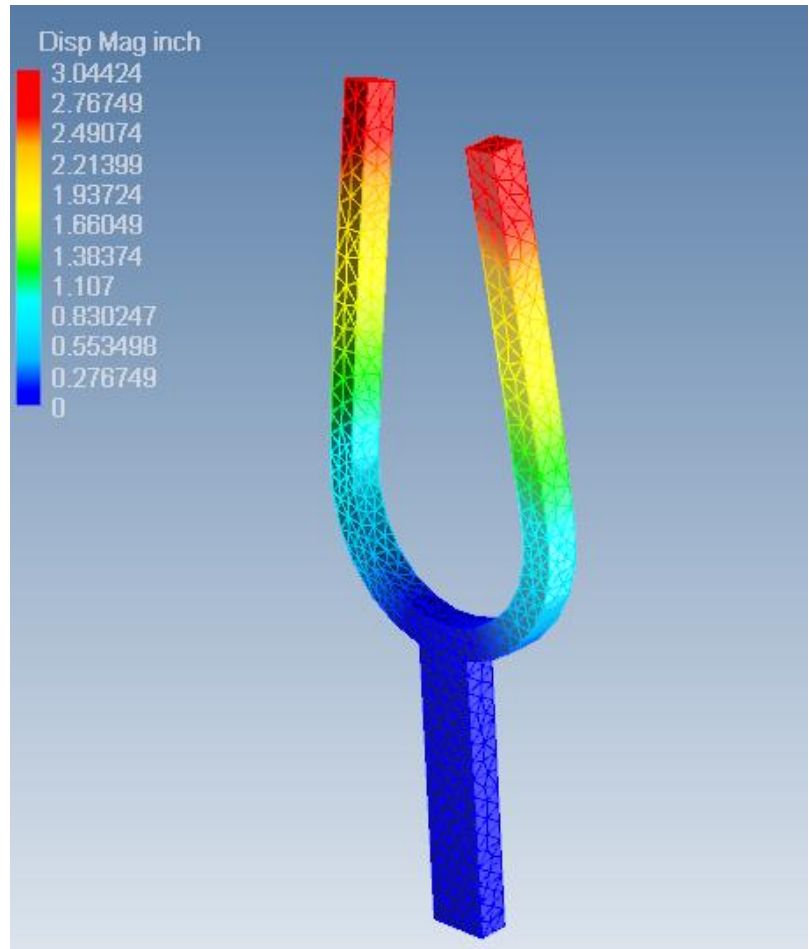
## Tuning Fork Model

The tuning fork model demonstrates how to perform modal analysis (harmonic vibration mode), buckling analysis, and frequency domain analysis. These three analysis functions are the most commonly used in structural dynamic design and in instability checking.

Modal analysis extracts the fundamental vibration modes of the model. These are the frequency a part should avoid during operations as it will create resonance responses. Frequency domain analysis can analyze a load or constraint at the given range of frequencies to determine the dynamic amplification load effect, and as demonstrate, if you excite a part over a range of frequencies containing the natural vibration modes, it will produce higher dynamic amplification response and could possibly damage the part. To see the effect, in the frequency domain Result section, select the Node inquiry, select a node around the loading source, and plot the displacement magnitude versus frequency to see the peak behavior.

Buckling analysis is a commonly used design instability analysis. It requires that you place a loading or constraint on the model, then compute the buckling factor(s) based on the loading status. The buckling factor (shown as "LoadFactor" in Result) is the factor scale of the applied loading/constraint. It informs the user that under the indicated factor of load/constraint, the system can go into the mode of instability buckling, and such factored loading state should be avoided or suppressed by adding more constrain to prevent such buckling instability.

Suggested what-if analysis: change the stiffness or mass to see the sensitive of the modal frequency; apply lateral constrain to the buckling analysis to see how it changes the buckling factor; use finer frequency increment in the frequency domain analysis to see how the displacement changes around the computed vibration mode.



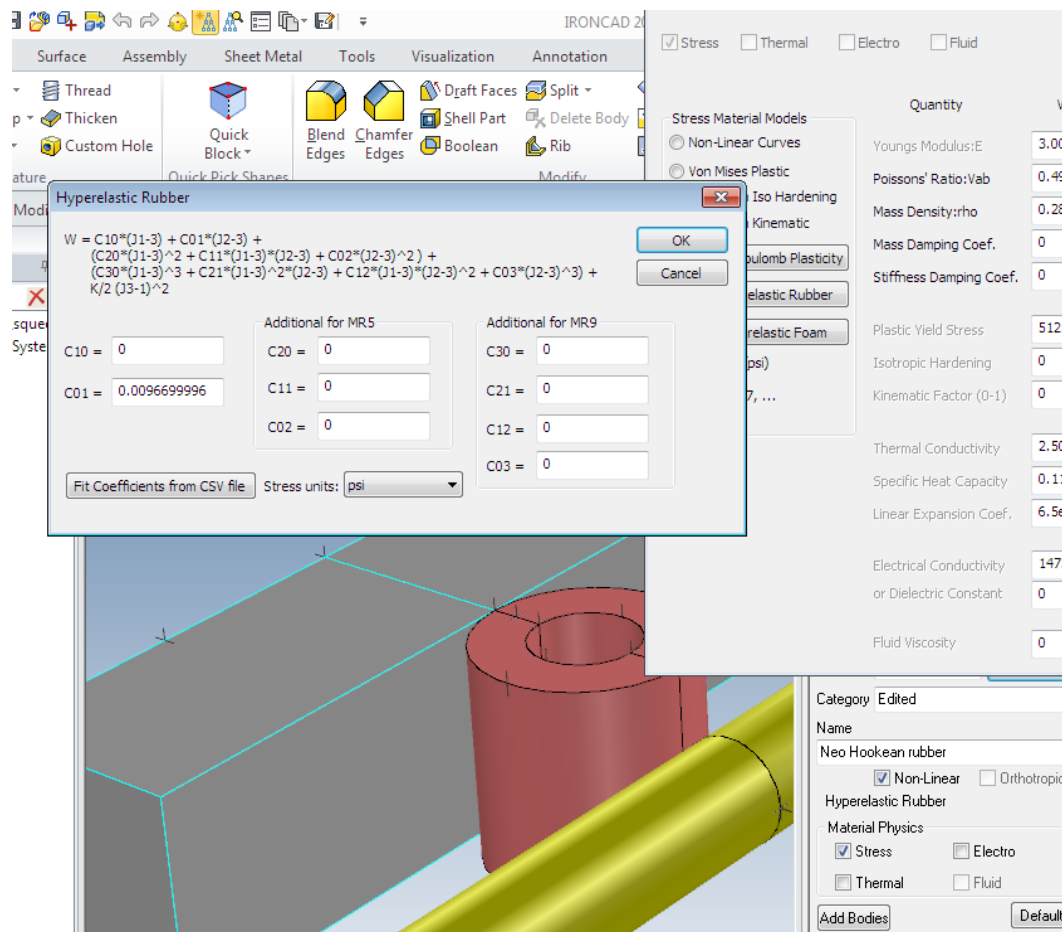
## Rubber Squeezing Model

The rubber squeezing model shows how to use the incompressible hyperelastic material model. The model simulates an experiment of squeezing a hollow rubber tubing against an incompressible rubber block of the same material using a rigid steel bar.

To use the hyperelastic material, the user can enter the experimental stress-strain curve, and the hyperelastic material model input tool will automatically compute the needed the general 9-coefficient Mooney-Rivlin model coefficients. In this case, we have entered the coefficient from an experiment based on a Neo-Hookean single coefficient model.

Since the final step of squeezing will compress element shape a lot and increase the nonlinearity, we refine the load step to half the initial increment at the half way into squeezing. We also show how different mesh refinement can be used to increase the simulation accuracy. In many situations, if you refine the mesh, it also requires to reduce the time/load step size.

Suggested what-if analysis: change the mesh size to see the sensitivity of the result; Increase/decrease the step size to see how program succeeds and fails after automatic step size cut down;

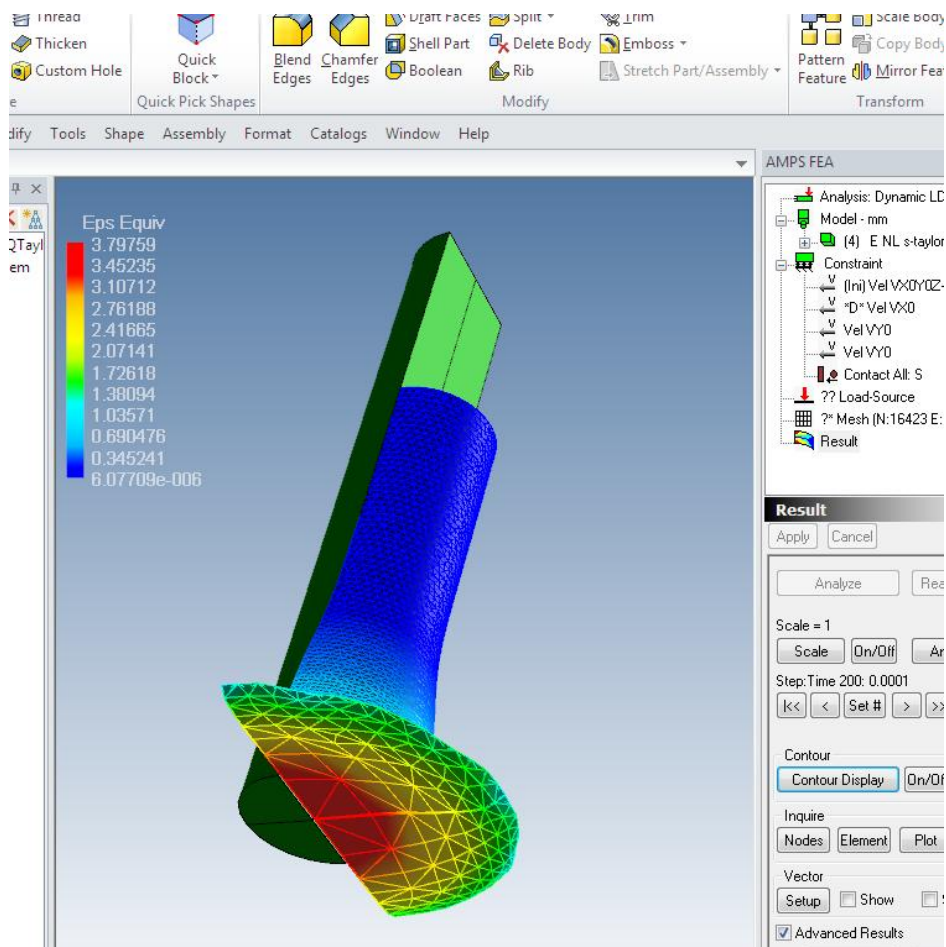


## Taylor Bar Impact Model

The Taylor bar impact model shows to perform a high speed dynamic impact analysis using a rigid contact plane with automatic contact analysis.

It is a seem-to-be simple but advanced nonlinear large deformation plasticity benchmark. The bar has an initial length of 32.4mm with an initial radius of 3.2 mm, Young's modulus= $1.17 \times 10^{11}$  Pa, Poisson's ratio=0.35, initial yield strength= $4.0 \times 10^8$  Pa, hardening modulus= $1.0 \times 10^8$  Pa, and mass density of  $8.93 \times 10^3$  Kg/m<sup>3</sup>. The bar has an initial impact speed of 227m/sec to a rigid frictionless wall. Initial time step of  $5.0 \times 10^{-7}$  seconds is taken for 200 steps.

Suggested what-if analysis: change the mesh size to see the sensitivity of the result; Increase/decrease the step size to see how program succeeds and fails after automatic step size cut down; Change the default Sefea Formulation and use the standard formulation; use the 2<sup>nd</sup> order TET10 element and check the much longer runtime and the increased iterations/instability.





# Mutltiphysics for Ironcad Demonstration Models (Rev. 1.0)

## Pool Game Model

The pool game simulation is based on an actual children's pool table so the size is smaller than the actual one to speed up the analysis. It demonstrates the automatic contact feature in IronCad Multiphysics simulation, and it also demonstrates the gravitational loading effect, multi-material setup, grouping, and the use of foam material for the pool table sides.

The model is setup to run as realistic as possible, but with the least amount of size. In the dynamic analysis, the traditional Courant step size criterion requires that the time step size must be small enough so the stress wave propagation (or dynamic impact wave) should not be traveling beyond the smallest mesh size, and the element traveling speed should also follow the similar rule. The stress wave speed is roughly equal to the squared root of the stiffness/mass ratio, and the theoretical allowable time step size will be very small. Based on advanced algorithm and formulations, the demonstrated step size of 0.005 is much larger than the theoretical limit for practical design analysis purpose.

In order to achieve more realistic behavior, the pool balls must have some frictional effect for realistic responses, and Coulomb friction coefficient of 0.02 is used. The pool table and the foam sides also have been modeled with a 1% damping to simulate the damping behavior happens in real game due to the pool table/side cloth surface. To reduce the result output file, the model only outputs the time step result at every other step (in Analysis/Advanced Optional controls).

Suggested what-if analysis: change the cue ball initial direction and see the response (till a ball is dropped and fall to the ground); increase the time step size and see how the program automatically cut down the time step to the allowable minimum step size; increase the pool ball mesh size so you can increase the time step to see how fast a one-second pool game motion can be achieved; change the damping ratio of the table and the side to see how it impact the behavior.

