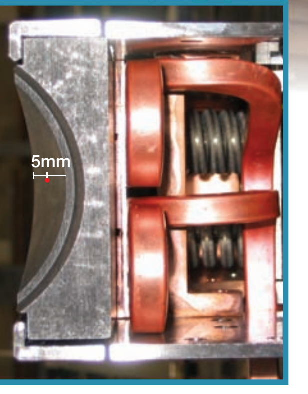
Glass Jaw

Simulation helps to solve collimator jaw design problem in the Large Hadron Collider.

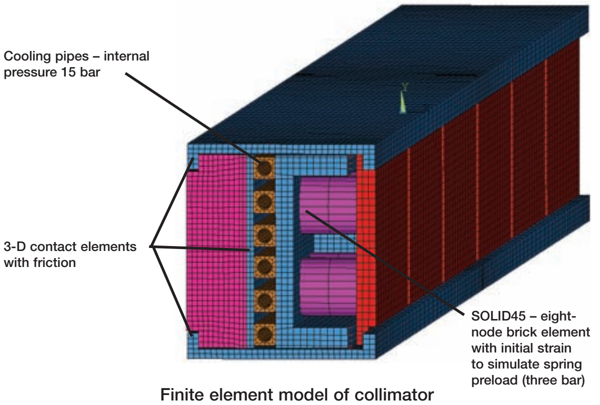
By Alessandro Bertarelli and Alessandro Dallocchio, Engineering Department, Mechanical and Material Engineering Group, European Organization for Nuclear Research, Geneva, Switzerland



Collimator cross section

The Large Hadron Collider (LHC) at the European Organization for Nuclear Research (Conseil Européen pour la Recherche Nucléaire, or CERN) is the world's largest and most powerful particle accelerator. Two beams of subatomic particles called hadrons — either protons or lead ions — travel in opposite directions inside the circular accelerator, gaining energy with every lap. The LHC's collimators scrape away particles that have gone slightly off track to prevent damage to the highly sensitive superconducting magnets. But the collimators need to be able to withstand an error that might pound them with a substantial fraction of the beam itself — considering that the beam could melt almost 1 ton of copper.

When researchers tested the LHC collimator prototype with several shots at different beam intensities, they discovered that the carbon–carbon collimator jaw nicely survived the impact, but its metal support suffered a permanent deflection strong enough to put the collimator out of action. CERN staff immediately set to work to understand what caused the problem and how to correct it.



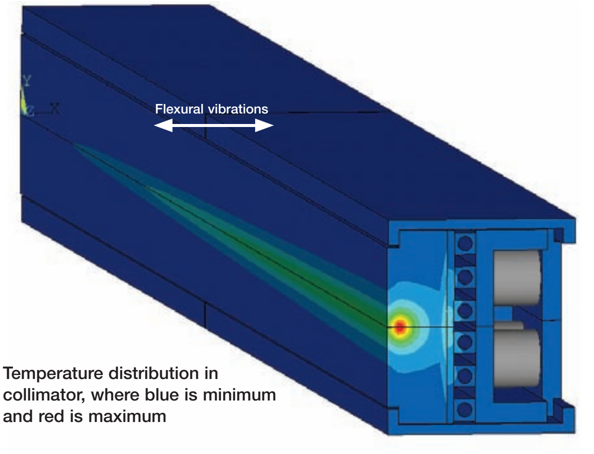
Finite element model of collimator

Sited near Geneva, Switzerland, the LHC is being exploited by physicists to study the smallest known particles that are building blocks of all things. The team uses the LHC to re-create conditions that existed just after the big bang by colliding two beams head-on at very high energy. The target of study is the Higgs boson, a theorized but not yet discovered particle that may help explain the masses of various subatomic particles. The LHC is being used to seek out dark matter that is believed to make up 96 percent of the mass of the universe. Scientists hope to discover the differences between matter and antimatter to help in understanding why so little antimatter is left in the universe even though the big bang is believed to have produced equal amounts of matter and antimatter.

The LHC uses a stored energy of 360 MJ per beam. This is two to three orders of magnitude above what other proton colliders can handle. CERN's collider is contained in an underground circular tunnel, 50 meters to 175 meters deep, with a circumference of 27 kilometers. The collider tunnel contains two adjacent parallel beam pipes, and each contains a proton beam that rotates in opposite directions around the ring. The parallel beams intersect at four points, forcing them to collide with each other. Some 1,232 dipole magnets keep the beams on a circular path, and an additional 392 quadrupole magnets focus the beams to increase the chances of collision at the intersection points.

The most critical elements of the collimators are the jaws, which are made of carbon–carbon composites, which encircle the beam and are designed to block any stray particles that separate from the beam. The jaws are supported by a brazed sandwich structure encompassing the main support bar, cooling pipes and interface plate.

The collimator is at risk when the beam is moved from one accelerator line to another. This needs to be done frequently because the beam is accelerated to maximum energy by moving it through a family of progressively more powerful accelerators. Kicker magnets are used to knock the beam out of its current line and inject it into the new line. If there is an error in the configuration of the kicker magnets, the beam might be steered too much in one direction. The collimator must be able to withstand accidents such as an injection error that may cause 3.2×10^{13} protons each with an energy of 450 gigaelectronvolt (GeV) to hit the collimator jaws. An electron volt is the kinetic energy gained by an electron when it accelerates through an electric potential difference of 1 volt.

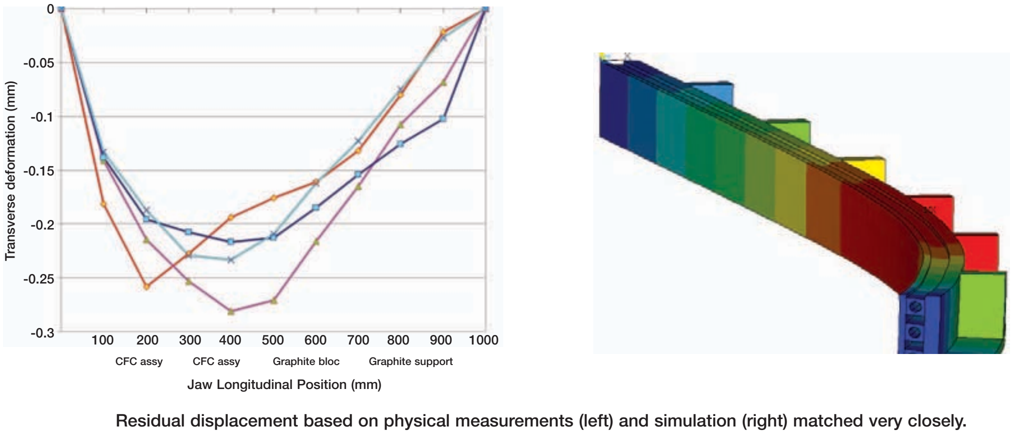


Temperature distribution in collimator, where blue is minimum and red is maximum

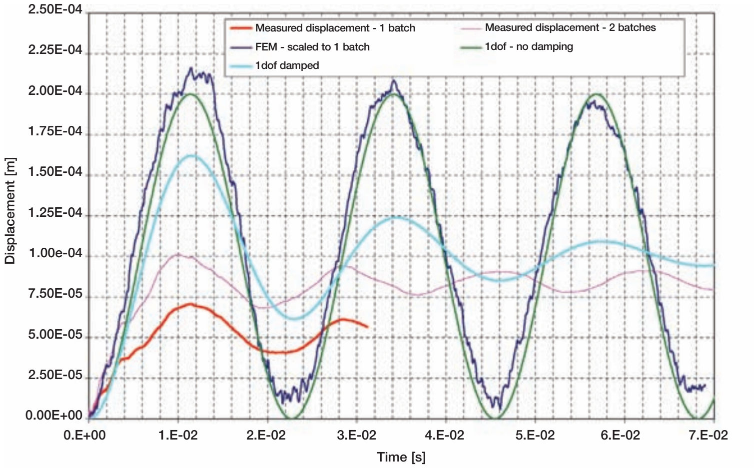
To validate the collimator design, engineers carried out tests using another CERN accelerator on a fully operational prototype of a collimator. The jaws were submitted to a series of impacts at 450 GeV over 7.2 μs. Measurements performed on jaw assemblies and metal supports revealed a permanent deformation of the metal support of over 300 μm. CERN researchers theorized that this deformation was caused by thermally induced vibrations due to very fast heating. Even when a structure is free to expand, when the heating process is faster than the typical stress relaxation time, material inertia prevents free thermal expansion, causing stress waves. Simulation was clearly required to better understand the problem.

Applications with such great mechanical complexity are typically addressed with explicit dynamics codes. But for this case, explicit analysis would have been very awkward because of the need for multiphysics integration and the complexity of the mechanical structure, which includes different materials that are held together with clamps. Since the stresses are well below the elastic modulus, there are no shock waves that would require an explicit dynamics code. The problem could instead be simulated using an implicit finite element model following the rules of thermo-elasticity. ANSYS Mechanical implicit finite element analysis software provides multiphysics capabilities that integrate thermal and mechanical analysis. In this application, integration increases accuracy and reduces the amount of time required to prepare the model. ANSYS Mechanical also offers comprehensive element technology, an extensive library of material models, auto contact detection for assemblies and powerful solver capabilities.

The collimation block was modeled as a rectangular beam, simply supported at its edges. The energy distribution applied to the collimator was determined with FLUKA, a particle physics Monte Carlo simulation package. The beam energy was introduced to the finite element model in the form of a 3-D table using the HGEN command. Transient thermal simulation was used to calculate the temperature distribution as a function of time. Researchers performed structural dynamic analysis by applying the temperature distribution as nodal loading at different time steps. Elastoplastic analysis used the multilinear kinematic hardening model for metallic components. The team developed special algorithms to apply a temperature distribution changing in time and space at different substeps of the elastic–plastic analysis. The integration time step of 0.1 μs was based on the



Residual displacement based on physical measurements (left) and simulation (right) matched very closely.

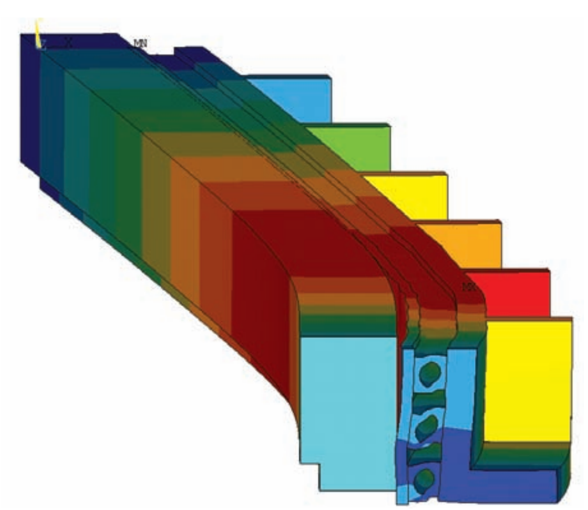


Simulation predictions correlate well with measured deflection.

preliminary analytical estimation to avoid numerical damping. Finally, static analysis evaluated residual plastic deformation.

The simulation results included the temperature rise throughout the structure, making it possible to use simple formulas to predict plasticization. The maximum stress is well above the proportional limit of copper, so the thermal shock generated plastic strains, as theorized by the researchers. The largest residual plastic strains of 0.12 percent were seen in the 3–mm thick copper plate. These strains are eccentric with respect to the neutral axis of the metal support, leading to permanent deflection away from the beam axis.

The simulation results matched the physical measurements remarkably closely. When comparing the physical measurements in the line graph to the simulation predictions in the contour plot, the transverse residual



Deflection was substantially reduced in the new design.

deflection of 350 μm predicted by the simulation closely matched the measured value of 300 μm. The dynamic response predicted by the simulation also correlated well with laser Doppler vibrometer measurements. The simulations are higher in magnitude than the physical measurements because the simulations do not take damping into account.

The study team, made up of senior and junior engineers, fellows, and students, used the validated simulation model as the primary tool to solve the problem. CERN researchers evaluated a number of different geometries and materials to determine their impact on the simulation of the support structure. Using these results, they decided to modify the jaw assembly series design by changing the thin plate material from OFE-copper to the higher yield strength Glidcop® material. Glidcop is a family of copper-based metal matrix composite alloys mixed primarily with aluminum oxide ceramic particles. The addition of small amounts of aluminum oxide greatly increases the copper's resistance to thermal softening and enhances elevated temperature strength.

An updated model of the series jaw assembly including Cu-Ni pipes and Glidcop support beam and thin plates showed that deflection was reduced from 300 μm to 16 μm, which was within acceptable limits. This figure was confirmed by experimental tests carried out on a second prototype of the collimator. The use of simulation in this application made it possible to rapidly diagnose the problem and develop an acceptable solution while eliminating the need to build additional prototypes, excepting the final design. ANSYS Mechanical software played a key role by providing the full range of physics capabilities needed to accurately simulate a very complex problem.