Finite Element Methods for the Thermo-mechanical analysis of the Phase I Collimators

LHC Collimation Project CERN

Workshop on Materials for Collimators and Beam Absorbers

3-5 Septermber, 2007

CERN Geneva

Alessandro Bertarelli[1]

Alessandro Dallocchio[1,2]

[1]TS department – Mechanical and Material Engineering Group
 CERN, Geneva
[2]Mechanical Engineering Department– Politecnico di Torino,
 Italy

Outline

- Introduction

- Application of the Thermal load

- Coupled Thermo-structural analysis (Fully elastic)

- Sequential Fast-transient thermo-structural analysis (elastic-plastic)

- Conclusions

Collimator Main Features

Overall length: 1480mm
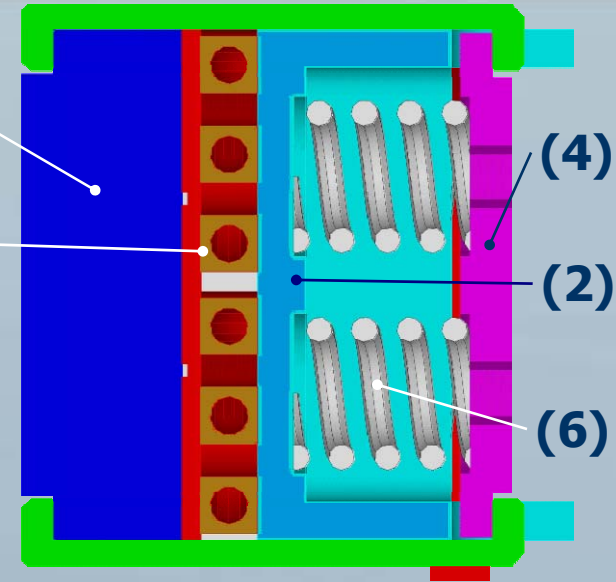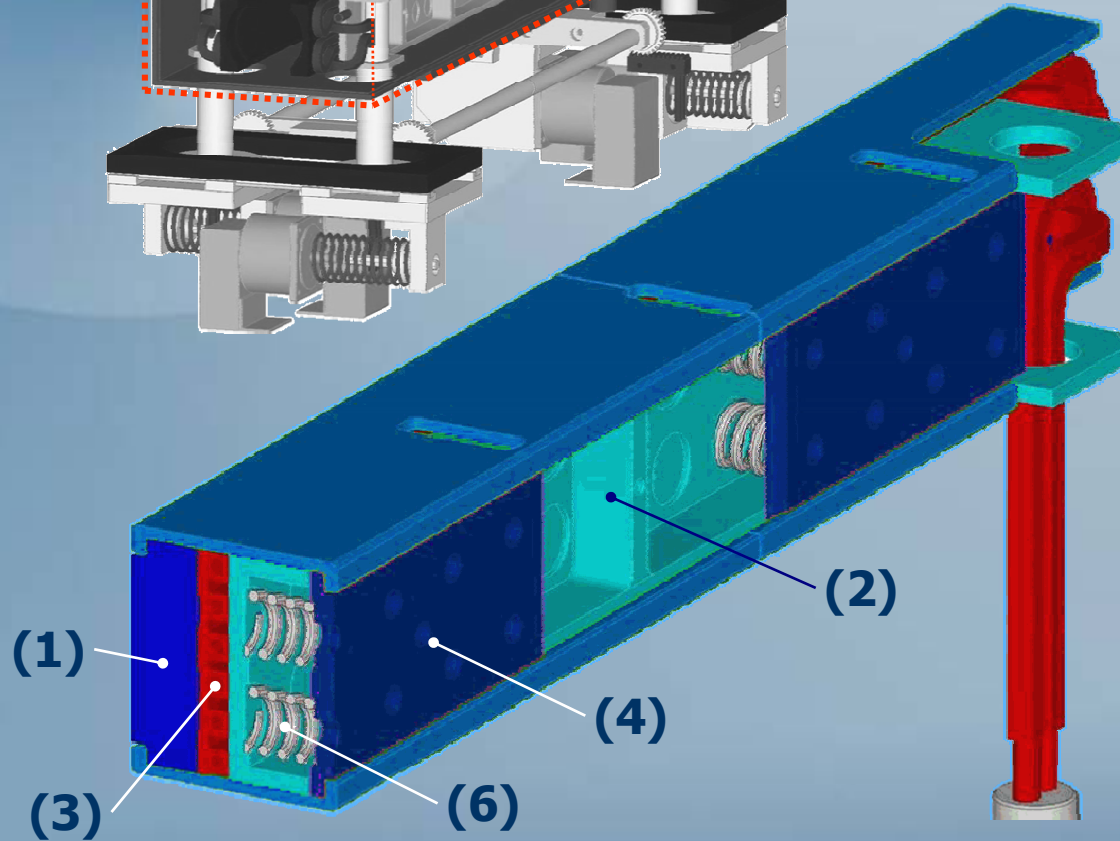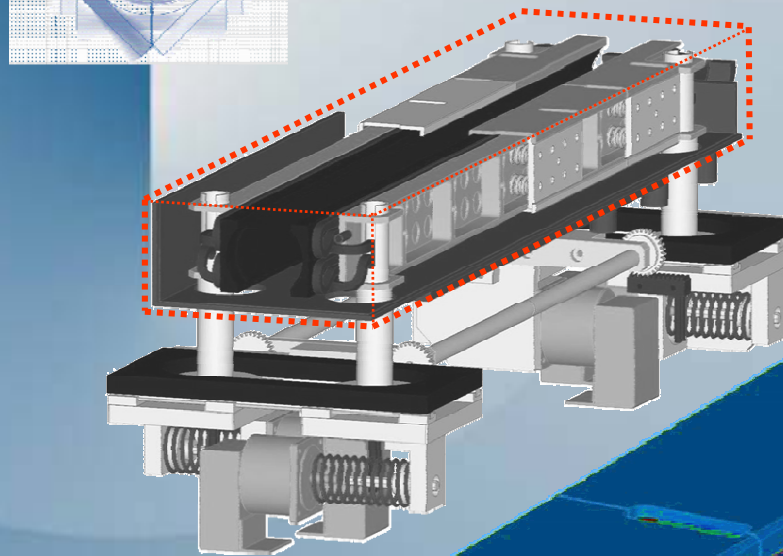Tank width: 260mm

Actuation system

Collimator assembly

Collimator tank

Beam axis

Collimator Main Features

Jaw Bloc Assembly

(1) Collimating Jaw (C/C composite)

(2) Main support beam (Glidcop)

(3) Cooling-circuit (Cu-Ni pipes)

(4) Counter-plates (Stainless steel)

(5) Clamping plates (Glidcop)

(6) Preloaded springs (Stainless steel)

Objectives of the Thermo-structural analyses

- **Verify heat loads are effectively evacuated without attaining excessive temperatures**

- **Verify that good thermal contact between C/C jaw and metal support is ensured**

- **Verify that thermal deformations are kept to a minimum in steady-state and transient regimes**

- **Verify the jaw and metal support survive to thermal shocks (⇨ See also A. Dallocchio's talk)**

To meet all these requirements, given the complexity of the design and physics (coupled analysis, contacts, preloads, convection, thermal loads etc.), a general use, comprehensive F.E. code was required ⇨

Ansys Multiphysics

- Introduction

- Application of the Thermal loads

- Coupled Thermo-structural analysis (Fully elastic)

- Sequential Fast-transient thermo-structural analysis (elastic-plastic)

- Conclusions

Application of Thermal Loads

- **Step1** Thermal load from Fluka files (GeV/cm³/p) are imported in Mathcad, rearranged and organized in a 3D matrix (XYZ) via a dedicated algorithm
- **Step2** 2D matrices (XY) at relevant longitudinal stations are extracted

5-columns, sequentially swept, Fluka input file

TCSGA6L1_FlukaData.dat - WordPad

File Edit View Insert Format Help

Courier New | 10 | Western | **B** *I* <u>U</u>

```
      LHC - IR7 Betatron cleaning insertion
      ***** Sum file *****
    710000.0            710000
           3
    TCPB6L1
          10              208
   -10.00000        10.00000             100   0.2000000
   -7.000000        7.000000              70   0.2000000
   -75.00000        75.00000             300   0.5000000
  F   0.000000        0.000000       1.0000000E+30
    5.6436477E-08    4.4016158E-08    7.1145813E-08    8.3932797E-08    9.0214307E-08
    5.7003756E-08    2.0147061E-07    4.2029851E-07    1.9455480E-07    1.4351802E-07
    9.9177754E-08    9.9034601E-08    1.1851952E-07    8.5688200E-08    2.0340570E-07
    2.2698151E-07    1.0822042E-07    1.9208935E-07    7.0475885E-08    8.4949761E-08
```
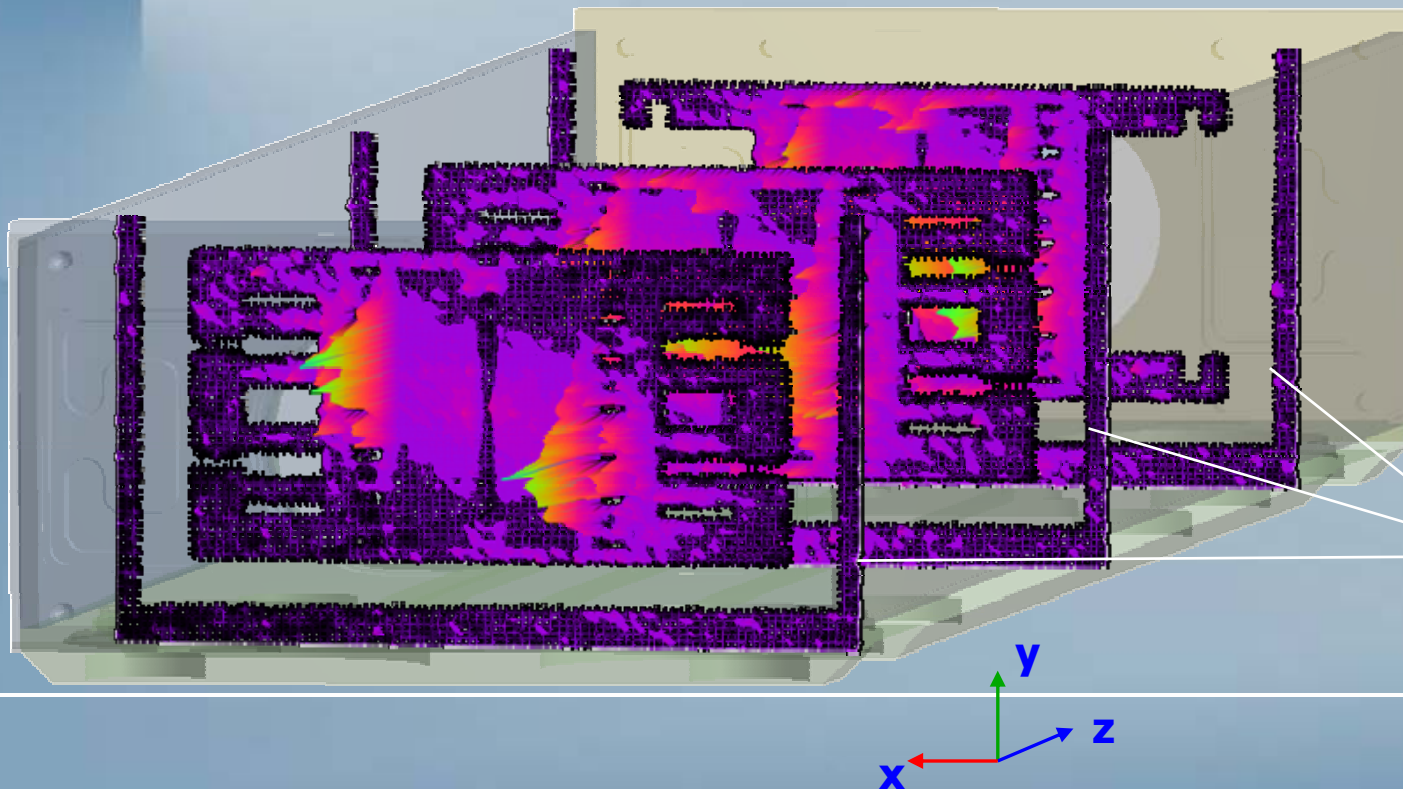
from Fluka) at a given longitudinal coordinate (z=1000 mm)
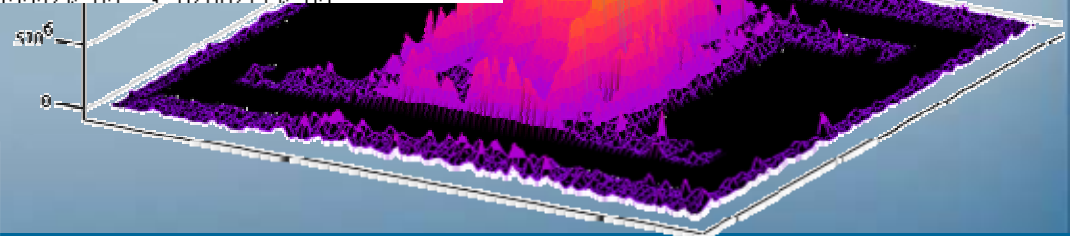
Application of Thermal Loads

Step3. These 2D matrices (for each station) are read into ANSYS as a 3D table and applied to the FEM model as Internal Heat Generation. Ansys then linearly interpolates values between each station.

ANSYS linearly interpolates heat loads between matrices in longitudinal direction

- Introduction

- Application of Heat Generation

- Coupled Thermo-structural analysis (Fully elastic)

- Sequential Fast-transient thermo-structural analysis (elastic-plastic)

- Conclusions

Thermo-mechanical analysis

A **fully coupled thermal/structural analysis** (i.e. the simultaneous resolution of temperature and displacement fields) is required since pressed-contact thermal conductance depends on contact pressure, which in turn varies with deformations .

$$h_c(P) = 1.49 \frac{k_S \Delta_a}{R_q} \left(\frac{2.3P}{E_G \Delta_a} \right)^{0.935}$$

CFC

Copper

P

P

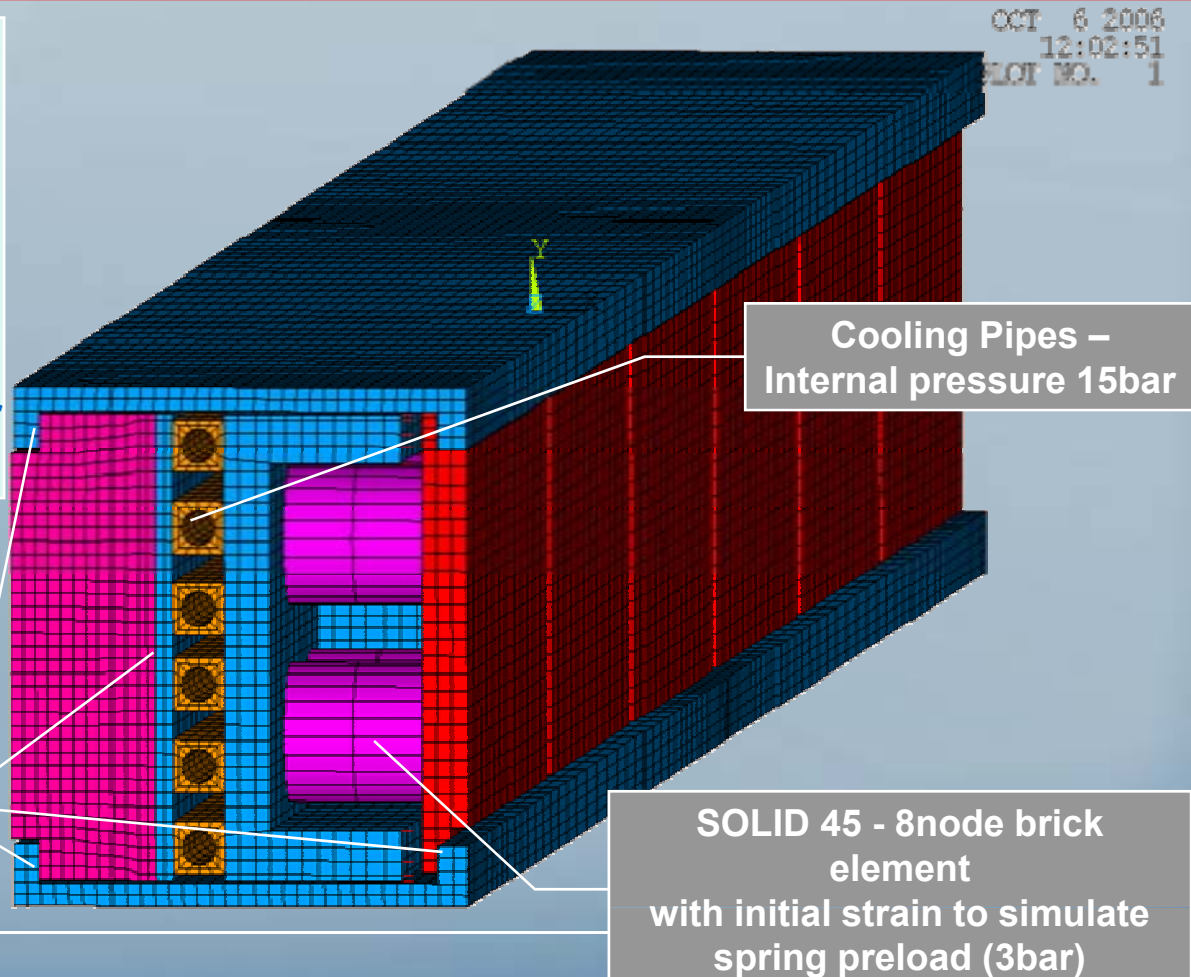Equation relating thermal conductance *h* to Pressure *P*
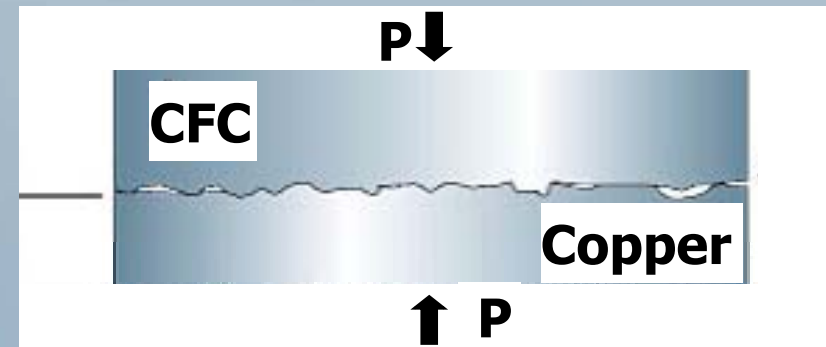
This relation, implemented in Ansys FE model, has been experimentally validated

Thermo-mechanical analysis

**Finite Element Model for 3-D Coupled Thermal-Structural Analysis
Steady-state (L.C. 1) and Slow-transient (L.C. 2)**

- **D.O.F..: ux, uy, uz, T**

- **SOLID5 - 8node Coupled Elements ~70000nodes**

- **Linear elastic materials with temperature-dependent properties**

- **3-D linear orthotropic model for C-C jaw**

- **3D Contact elements: CONTA173 - TARGET170   d.o.f.: ux, uy, uz, T**
- **Lagrange and penalty method (for precision and convergence)**
- **Contact Friction taken into account**
- **Thermal contact conductance *h* function of contact pressure *P***

Cooling Pipes – Internal pressure 15bar

SOLID 45 - 8node brick element with initial strain to simulate spring preload (3bar)

Thermo-mechanical analysis

Temperature - dependent material properties

Convection (12360W/m²/K) + inlet temp. (27ºC)

Deposited Heat Map (W/m³) as interpolated by Ansys from 3D Table

Simply supported extremities

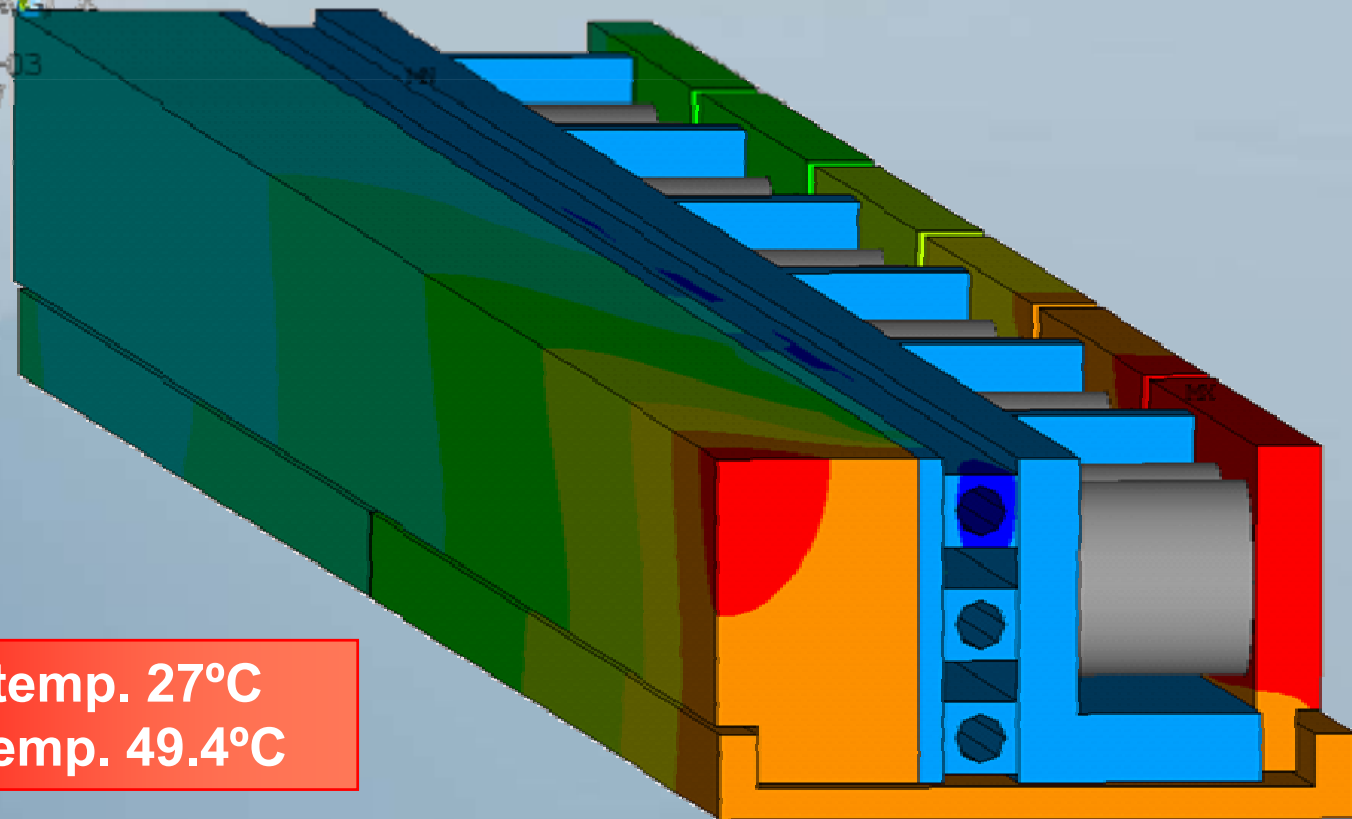0   488889   977778   .147E+07   .196E+07   .244E+07   .293E+07   .342E+07   .391E+07   .440E+07

Thermo-mechanical analysis

Thermal analysis – Load Case 1 (Nominal conditions) –
Steady state – Loss rate 8×10^{10} p/s – 4.5kW (Beam Lifetime 1h)

**Initial temp. 27ºC
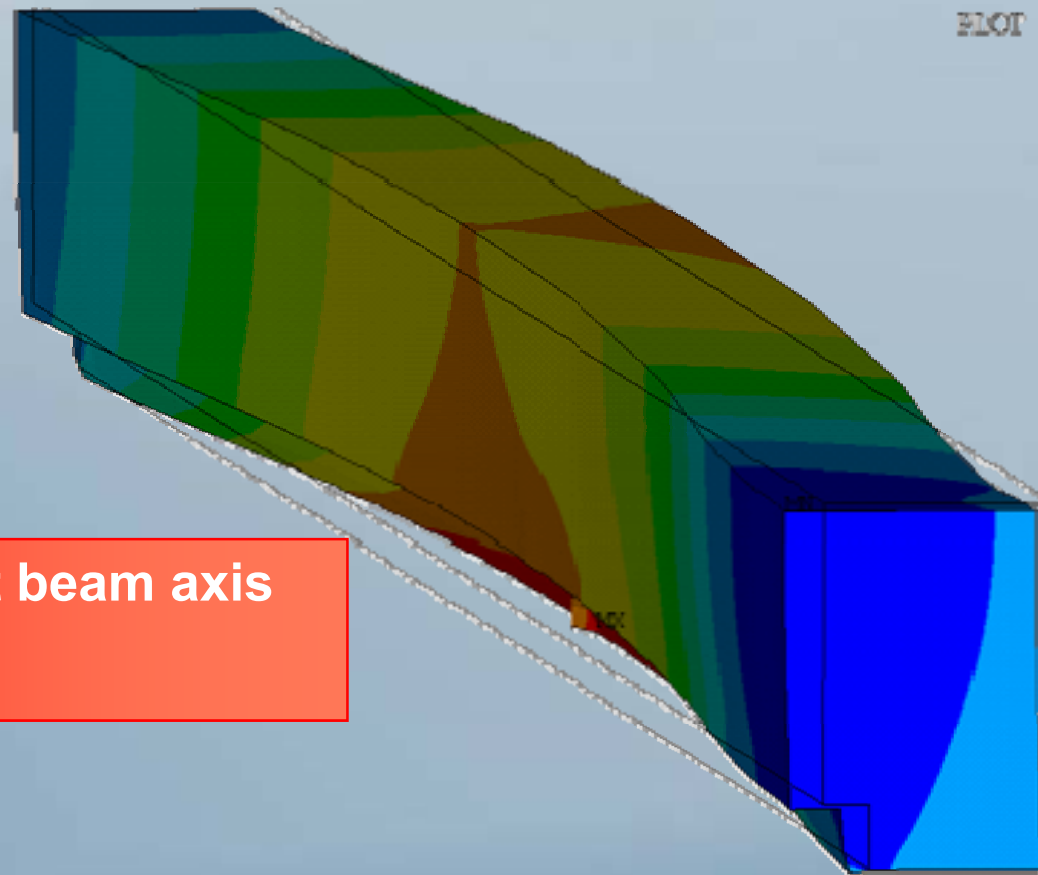Max. temp. 49.4ºC**

Thermo-mechanical analysis

Displacement analysis – Load Case 1 (Nominal conditions) – Steady state – Loss rate 8×10^{10} p/s – 4.5kW (Beam Lifetime 1h)

**Total deflection at beam axis**

$\delta_{Max} < 30\mu m$

Thermo-mechanical analysis

Thermal analysis – Load Case 2 (Nominal conditions) –
10s Transient – Loss rate 4×10^{11} p/s - 22kW (Beam Lifetime 12min)

1. **Quasi-static load step at L.C.1**

2. **10ms transient ramp from L.C.1 to L.C.2 (20 sub-steps)**

3. **10s transient analysis at L.C.2 (20 sub-steps)**

4. **10ms transient ramp from L.C.2 to L.C.1 (20 sub-steps)**

5. **20s transient analysis at L.C.1 (20 sub-steps)**

6. **Final quasi-static load step at L.C1**

Initial temp. from quasi-static analysis - Max. temp. after 10s 76.2ºC
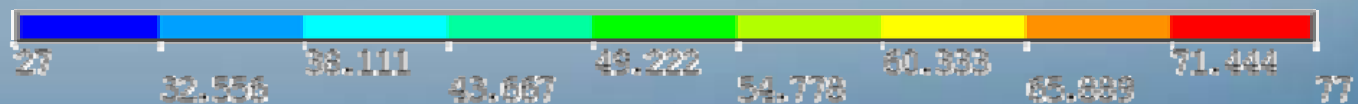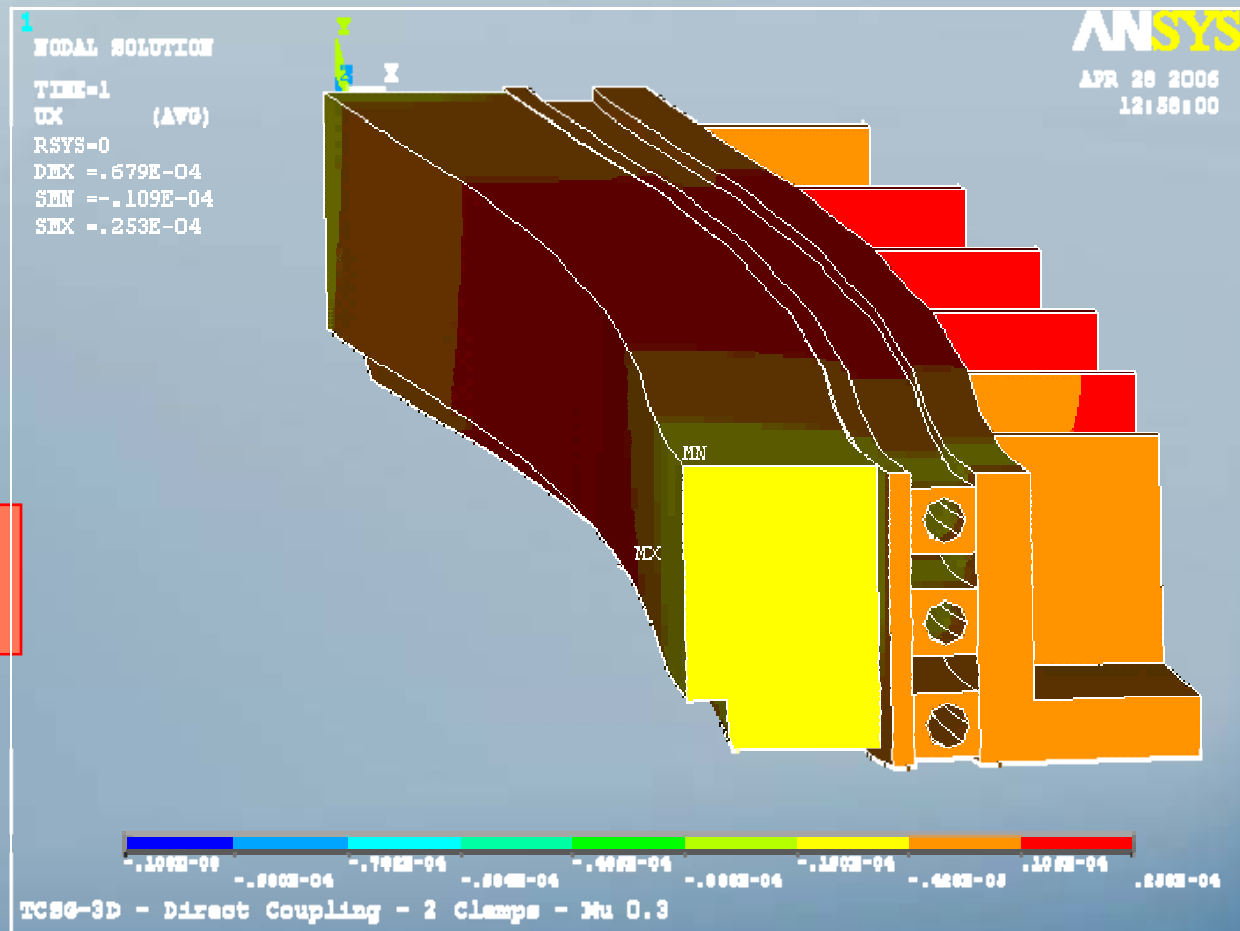
Thermo-mechanical analysis

Displacement analysis – Nominal conditions – Load Case 2
10s Transient – Loss rate 4×10^{11} p/s – 30kW (Beam Lifetime 12min)

Initial loss 8e10p/s
Max. deflect. ~20µm

Transient loss
4e11p/s during 10s

Max deflect. -108µm
Back to 8e10p/s

Outline

- Introduction

- Application of Heat Generation

- Coupled Thermo-structural analysis (Fully elastic)

- Sequential Fast-transient thermo-structural analysis (elastic-plastic)

- Conclusions

Simulation of Beam impact accidents

A fast-transient analyses are necessary to simulate accident scenarios: in particular 450 GeV Injection error test …

5 full intensity shots ranging from 1 to 5 mm, 7.2μs …
Each impact energy equivalent to more than ½ kg of TNT

5 mm

min ▮▮▮▮▮▮▮▮▮▮ max
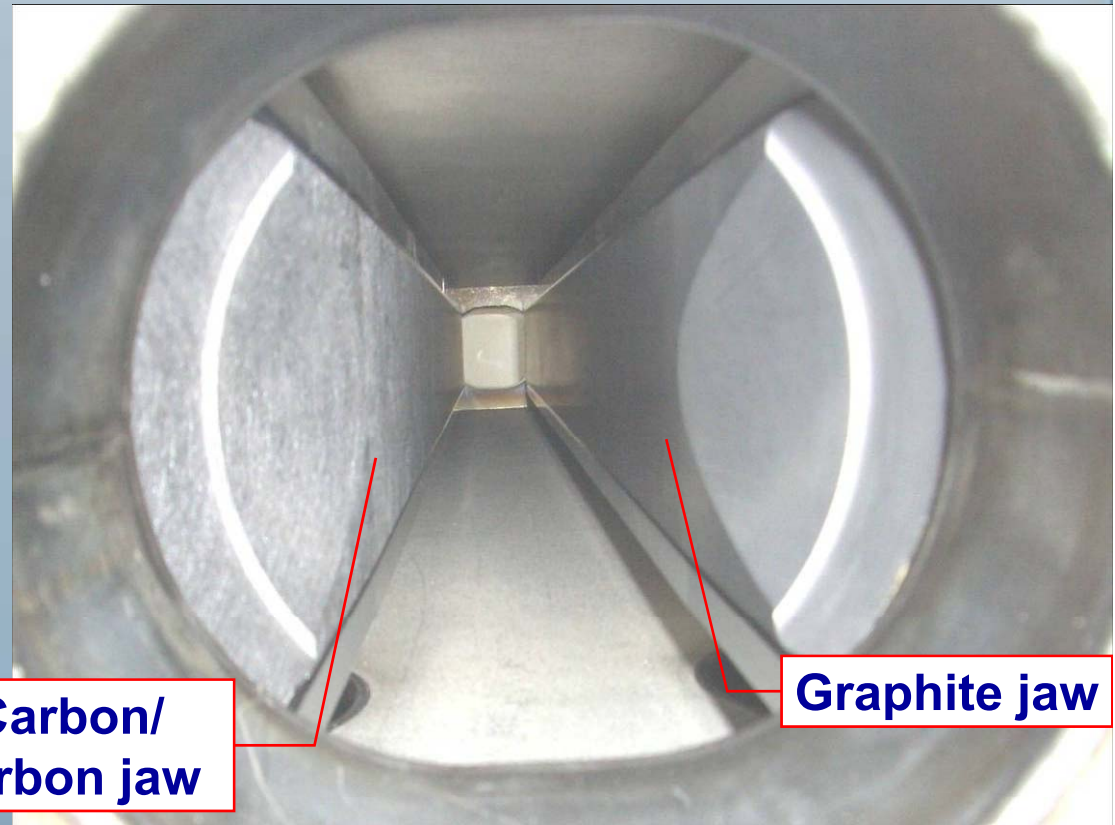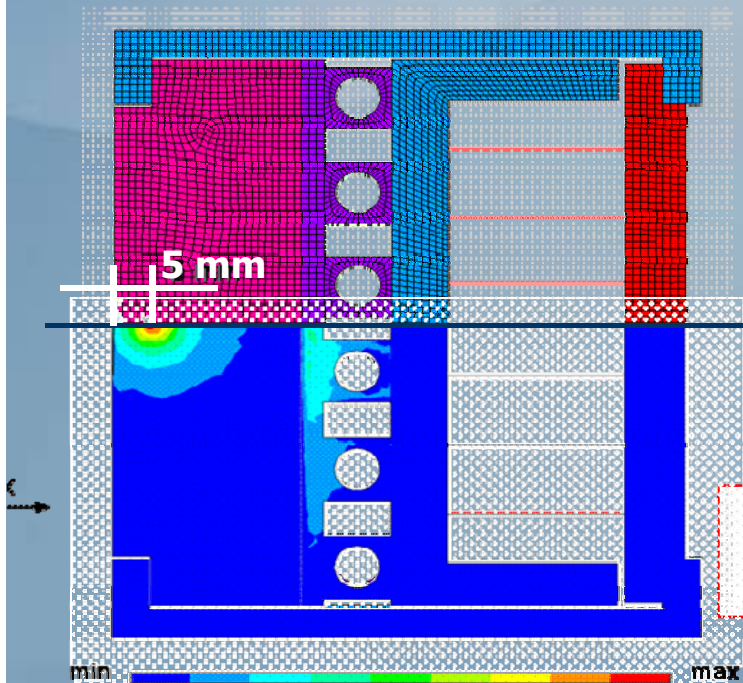
Carbon/carbon jaw

Graphite jaw

Sequential analysis

Can we perform these analysis with Ansys Multiphysics, an implicit-integration scheme Finite Element code?

- **Ansys fully coupled thermal/structural analyses can only be performed in the linear elastic domain**

- **Numerical analyses in the elastic/plastic domain are necessary to simulate possible permanent deformation of metallic parts in case of accidents (as those found after 2004 Injection error test)**

But ...

- **Accident cases entail fast energy depositions (a few μs or less)**

- **Given the typical thermal diffusion times ($\sim ms$ or more – $\tau_{diff}=L^2/\kappa$), heat conduction plays a minor role in case of very short time-scale analyses.**

- **Hence, thermal and structural analyses can be decoupled and sequentially solved with Ansys Multiphysics**
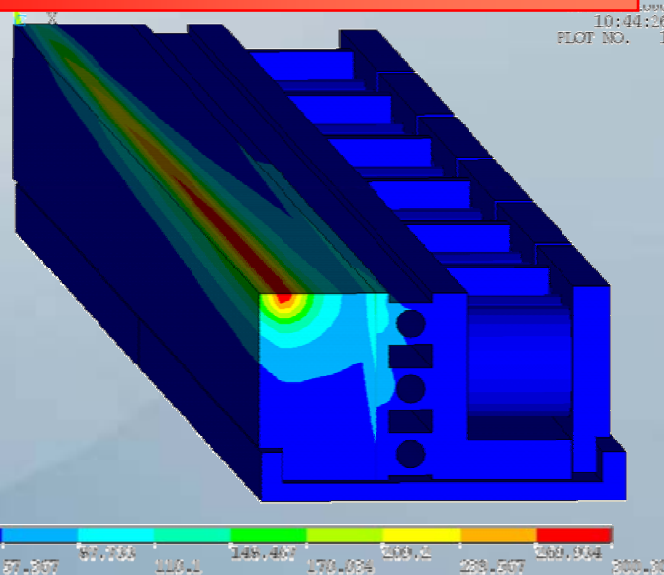
The price to pay ...

- **Be patient ... very short time-steps ($\Delta T < 0.9 L_{mesh}/c < 1\mu s$), at each step inversion of the stiffness matrix $\Rightarrow$ very long CPU times**
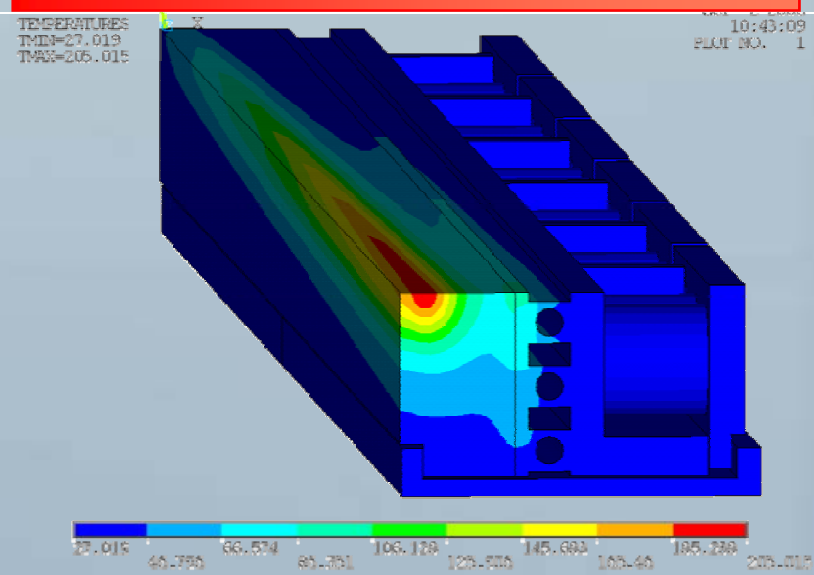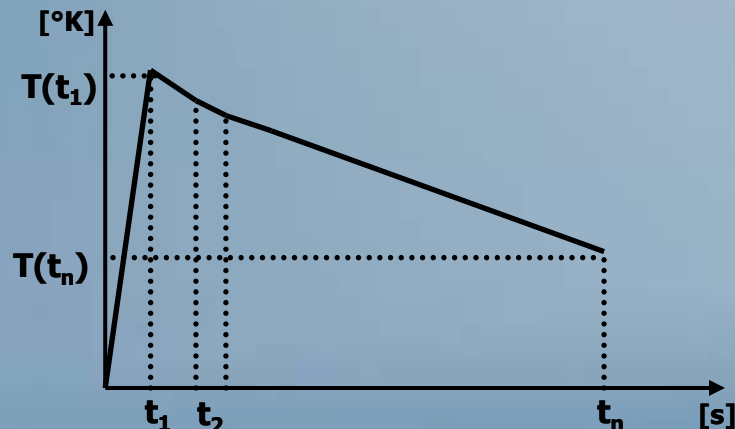
Thermal analysis
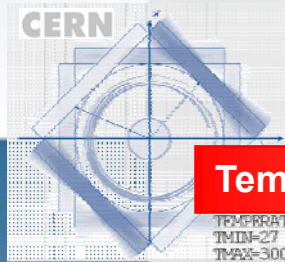
Temperature distribution at time t = 7.2µs

Temperature distribution at time t = 60ms

Temperature distribution as a function of time

- **Temperature distributions from thermal analysis were applied at different sub-steps (time)**
- **Ansys linearly interpolates between sub-steps loads so quite closely following actual temperature evolution**

Fast-transient structural analysis

Collimator jaw oscillates several ms after the beam impact - *Thermally induced vibrations* have been fully studied with a 3D FE model in the elastic-plastic domain (see A. Dallocchio's talk for more details**) .**
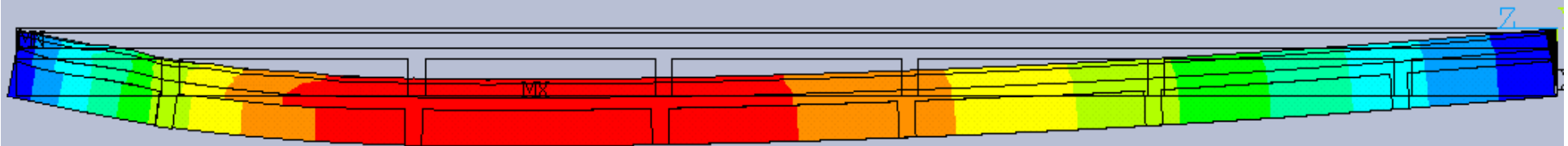
A multilinear-kinematic hardening plasticity model was used for metals

- Over a long time scale the dynamics effects (inertia) were turned off.

- A last static load step was necessary to calculate equilibrium between plastic residual stresses and elastic forces.

- Numerical results are in line with analytical prediction

-.290E-06		.329E-05		.687E-05		.105E-04		.140E-04
	.150E-05		.508E-05		.866E-05		.122E-04	.158E-04
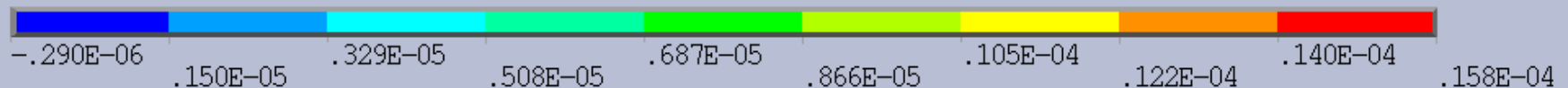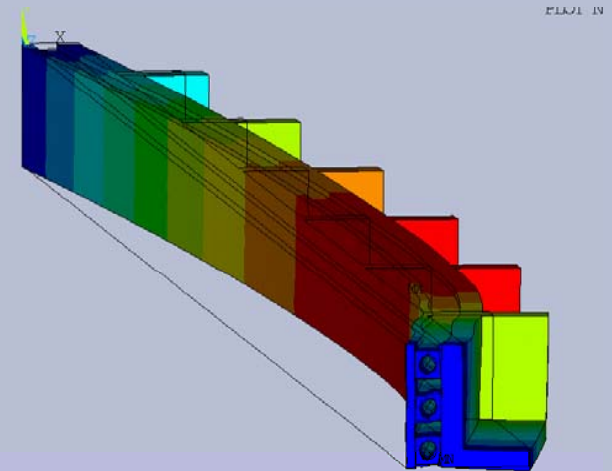
Predicted permanent deformation is not zero (because of limited plasticity on the CuNi pipes) but well within limits (16 μm)

Outline

- Introduction

- Application of Heat Generation

- Coupled Thermo-structural analysis (Fully elastic)

- Sequential Fast-transient thermo-structural analysis (elastic-plastic)

- Conclusions

Conclusions I

- In-depth thermo-structural simulations were performed on Phase I collimator models from the early stages allowing design optimization; variable thermal loads from FLUKA were applied making use of 3-D tables.

- Thermal/structural coupled elements used. On relatively long time-scales, it is not possible to separate the two domains (thermal conductance depends on contact pressure), making the problem intrinsically coupled.

- For contact elements (which include friction) the *Lagrange and Penalty* method was used allowing control of contact penetration.

- Studies of solutions accuracy, influence of mesh dimensions and time-steps were done.

- Comparisons with analytical calculations and experimental results were performed to validate FE models.

- Sequential fast-transient thermo-structural analyses were performed in the elastic-plastic domain to predict collimator behavior in case of beam impacts.

- Solutions allowed to study thermally-induced vibrations on the short time scale and predict residual deformations on the long time-scale.

- We believe Ansys Multiphysics can be successfully applied for the reliable solution of complex fast-transient thermo-mechanical problems, provided changes of phase are not occurring or stress level is well below material Young's modulus ($\Rightarrow$ hydrodynamic theory)

- The main implicit method drawback is long solution times imposed by very short time-steps (this can be offset by powerful hardware)