LASER BEAM DUMP DESIGN OPTIMIZATION USING FINITE VOLUME METHOD

SUPERLASERS FOR THE REAL WORLD

Picmausová H., Crha P., Macúchová K., Heřmánek J., Divoký M., Řeháková M., Mocek, T., HiLASE Centre, Institute of Physics AS CR, v.v.i., Za Radnicí 828 Dolní Břežany, ČR

INTRODUCTION

In a laser system, the role of a beam dump, as its name suggests, is to safely absorb the energy of the photons forming the beam. Safe and efficient beam dumps are essential for high-power systems such as BIVOJ, where a simple optical element would be unable to withstand the peak power of the laser. Specific configuration of materials is decided based on the power and wavelength of the laser beam. Beam dumps introduced in this paper are all cylindrical-shaped tanks with glass front lids, filled with flowing water as shown below. Using water as absorptive medium is given by fact that BIVOJ laser, for which the beam dumps are intended, operates mainly around 1550 nm – wavelength at which water has a marked absorption peak.

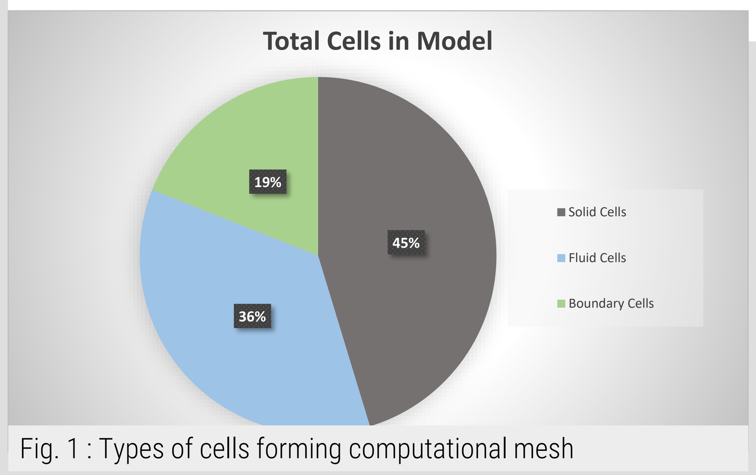


Fig. 1 : Types of cells forming computational mesh

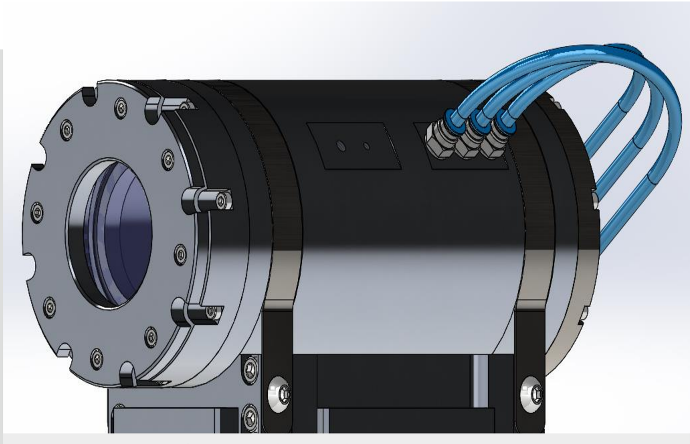


Fig. 2 : Origingal Beam dump design

NUMERICAL MODEL

The numerical simulations were run in SOLIDWORKS Flow Simulation Module using one of its two Finite Volume Method solvers. The solver uses a SIMPLE-like algorithm and a preconditioned generalized conjugate gradient method with incomplete LU factorization, and a structured mesh for the computational domain.
As opposed to their more popular unstructured and hybrid counterparts, structured meshes can easily be represented by two- or three-dimensional arrays in computer memory, thus making the solver highly efficient. They also allow for a much higher degree of control of the cell size and node placement, taking into account key areas such as sharp edges, narrow channels and solid-fluid boundaries within the CAD model. Mesh generation for these simulations yielded approximately 4.200.000 cells of three main types, as shown in Fig. 1.

ORIGINAL DESIGN

The original design, as seen Fig. 2, has three inlet nozzles with a 3 l/min flow rate, and a single outlet on the opposite side of the cylinder's diameter.
This set-up, while perfectly sufficient for its current purpose, offered several opportunities for design improvement. The flow inlet placed normally to the circumference created a flow field with marked pressure and velocity loss, while creating very little turbulence intensity inside the liquid volume. This resulted in a large portion of incident energy being absorbed by the body of the beam dump rather than the cooling water, as is apparent from Fig. 3 and flux plot in Fig. 4.

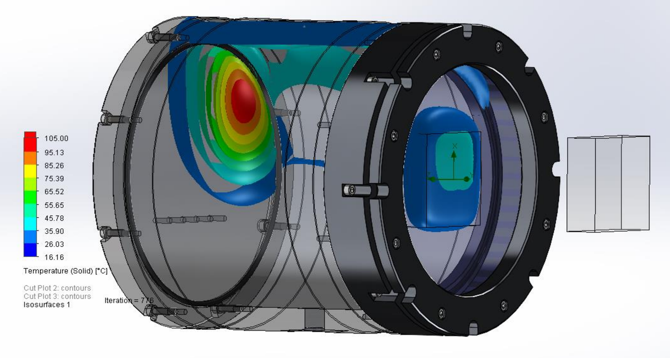


Fig. 3 : Isosurfaces of temperature distribution in original model

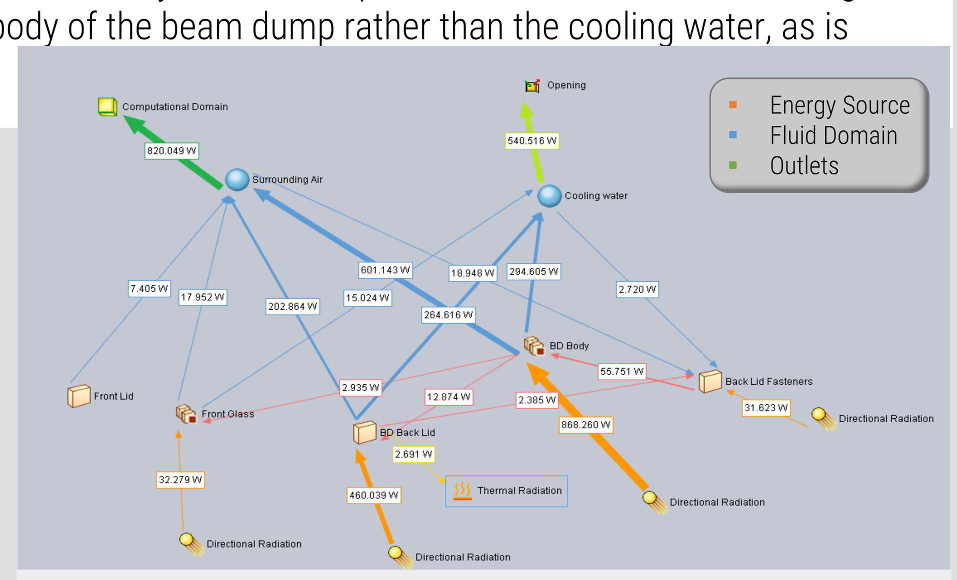


Fig. 4 : Flux plot scheme of heat flux within the original model

IMPROVED DESIGN

Keeping in mind these simulation results and possible future use of the component, a more efficient flow field set-up was required. The first step was to place both the water inlets and outlets tangentially to the beam dump body as depicted in Fig. 5.
This relatively simple modification resulted in higher vorticity in the flow field and improved velocity, as evidenced in Fig. 6.
Although this version of the model still lacks desired turbulence intensity for optimal cooling, it provides a much more desirable heat flux scheme as opposed to the original design.

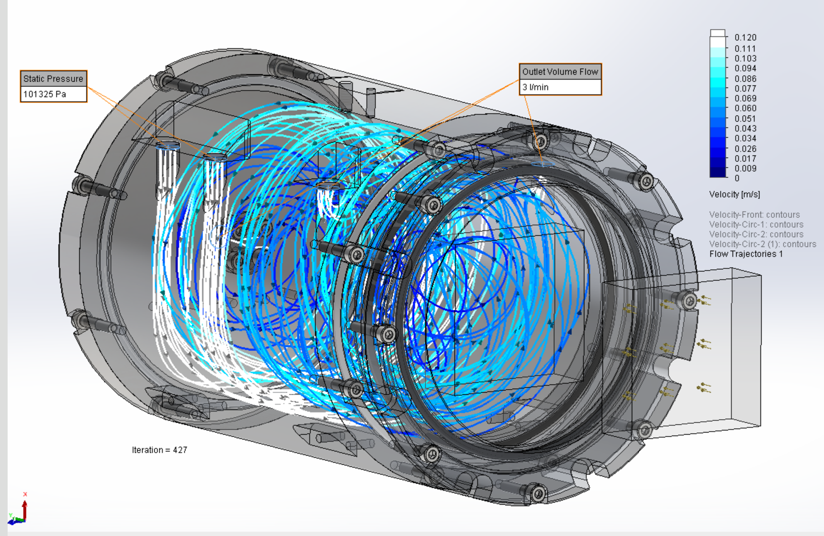


Fig. 5 : Flow field and boundary conditions on updated set-up

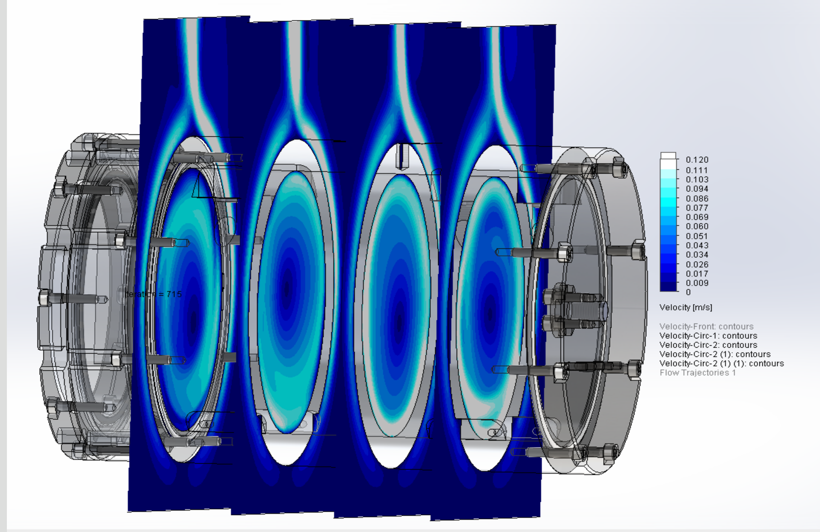


Fig. 6 : Velocity field cut plots on updated set-up

GENERATING TURBULENCE

Although both previous versions of the beam dump are sufficient in their cooling capacity, efficiency of any heat management flow field is improved with higher turbulence intensity. In a further modification, this was achieved by adding circular rods near the inner circumference of the beam dump body, while keeping the improved set up of inlets and outlets for the cooling water. This design, although the most technologically challenging to manufacture, has proven the most efficient in heat management, as reflected by the lowest maximum temperature of the front glass in Fig. 7, as well as the most power channelled away via cooling water and the least amount radiated into surrounding air – see flux plot in Fig. 8.

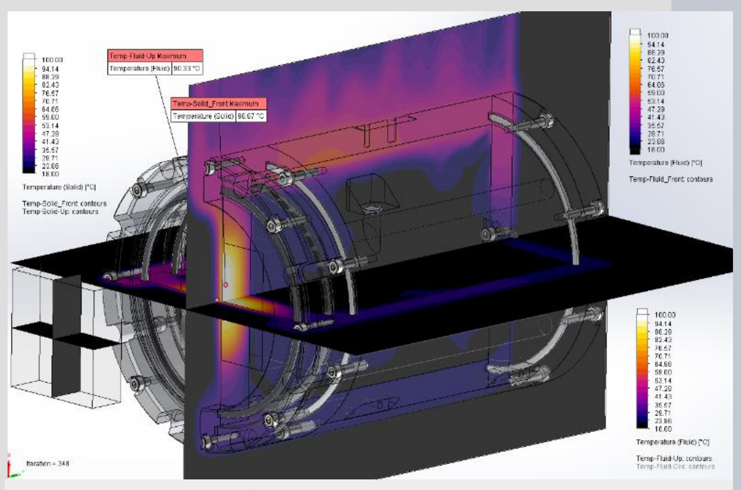


Fig. 7 : Temperature field plots on final set-up

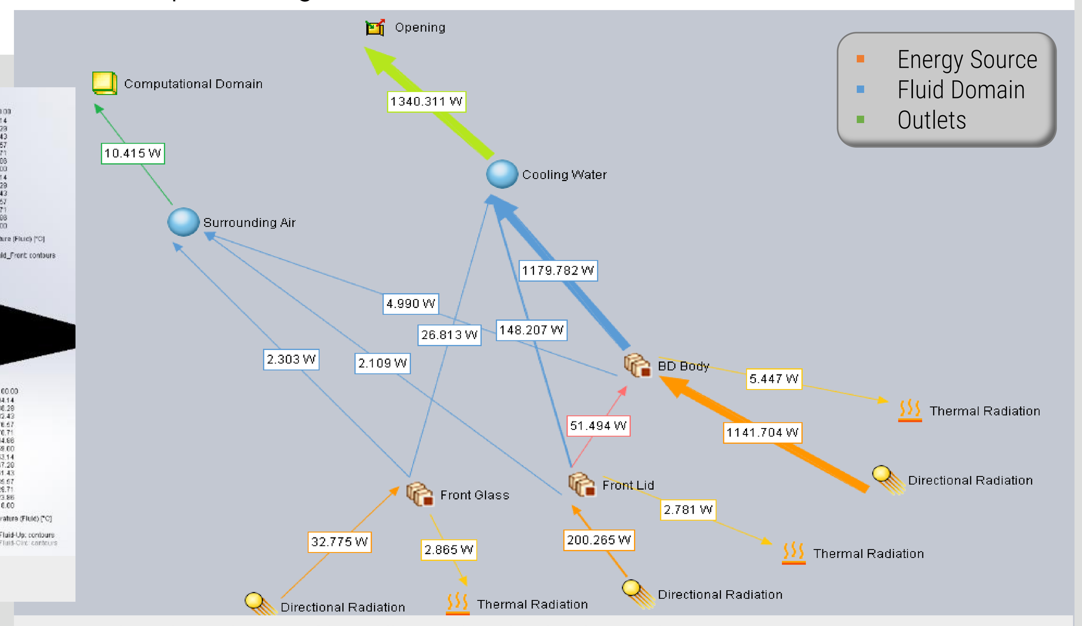


Fig. 8 : Flux plot scheme of heat flux within the final model

COMPARISONS

These improvements are a result of adapting design to create a turbulent flow field inside the beam dump. Clear comparison between the two newer designs in Fig. 9 and Fig. 10 show a marked difference in turbulence fields that resulted in more efficient heat management.

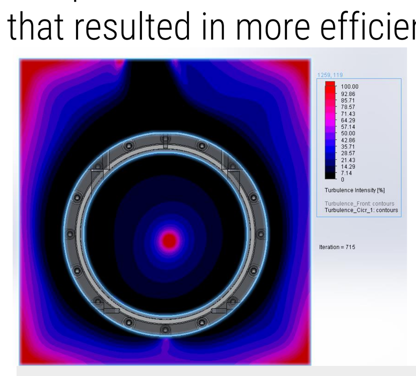


Fig. 9 : Turbulence plot – updated model

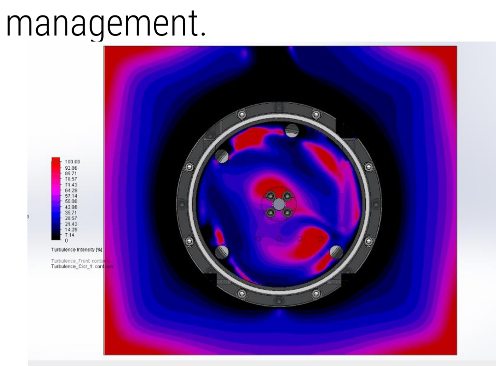


Fig. 10 : Turbulence plot – final model

All previous simulations were run as time-independent studies in order to maintain computation efficiency. When comparing similar designs, this approach improves speed without sacrificing accuracy. In real use however, the incident laser beam operates in pulsed mode. Therefore, another simulation was run to approximate real conditions with higher precision. Heat management-wise, this is a much more 'forgiving' mode, as the switching of the beam allows for much faster cooling of the components, as compared in Fig. 11, 12, and 13.

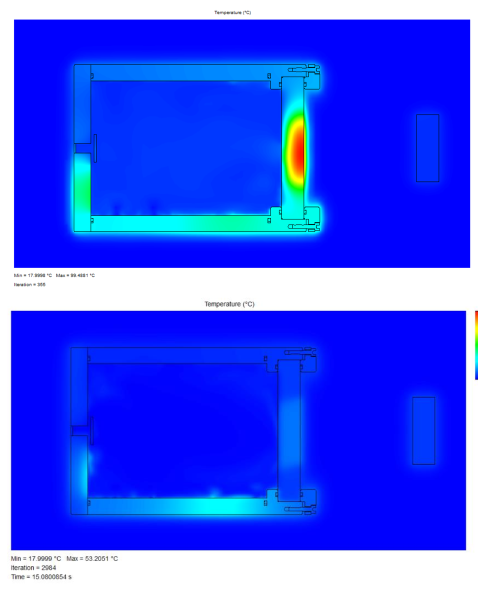


Fig. 11 : Comparison of temperature distribution in stationary and time-switched energy source simulations

Fig. 12 : Maximum and average temperatures in model – iteration dependent

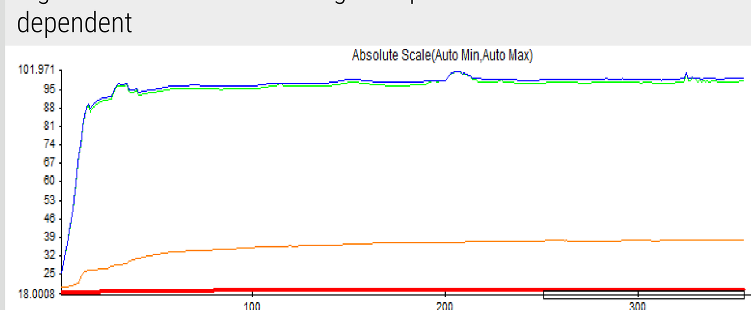
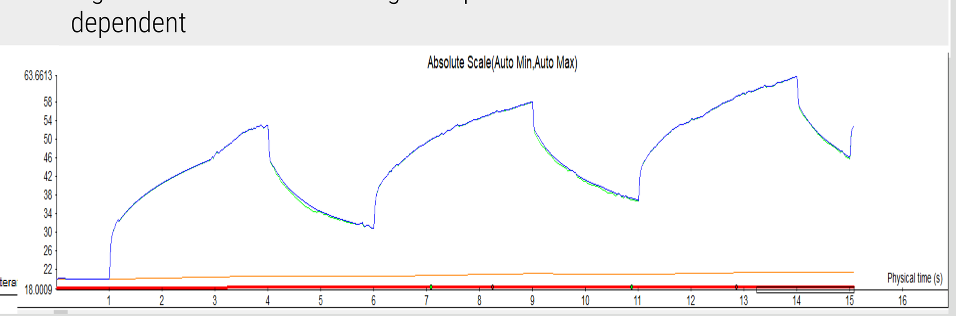


Fig. 13 : Maximum and average temperatures in time-switched mode - time dependent



CONCLUSION AND DISCUSSION

Time dependent studies are much more demanding on computational capability; therefore, time-dependent study was only run on the second version of the design. It shows the actual temperature is well under the tolerance scale for all components, and all three versions are sufficient for use in current projects. This paper demonstrates the possible improvements to the beam dump's design, aided by numerical experiments, for future use under more demanding conditions. Ensuring an efficient flow field with lower pressure and velocity losses and higher turbulence markedly improves cooling capacity of the set up. In future studies for higher volume flow rates and larger diameter beam dumps, the optimal shape, placement, and incidence angle of turbulence rods could be considered, as well as a possible comparison between solution convergence of different FVM and FEM solvers.

AKNOWLEDGEMENTS AND REFERENCES

- Sobachkin, Numerical Basis of CAD-Embedded CFD – SOLIDWORKS, Dassault Systemes, 2014

- This analysis would not exist without the dedication and helpful professional input of M. Divoký and the entire HiLASE ETS Design team – namely K. Macúchová, P. Crha, and J. Heřmánek, whom I would like to thank for their constructive contributions and help.

hilase

EVROPSKÁ UNIE
Evropské strukturální a investiční fondy
Operační program Výzkum, vývoj a vzdělávání

MINISTERSTVO ŠKOLSTVÍ, MLÁDEŽE A TĚLOVÝCHOVY

This project has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No 739573 (HiLASE CoE)

SOLIDWORKS

FZU