Mu2e Proton Beam Dump Heat Removal System

Italian Summer Intern Program 2024
Final Presentation - 26th September

Davide Venturini
MSc Nuclear Engineering
Università di Pisa and
Scuola Superiore Sant'Anna

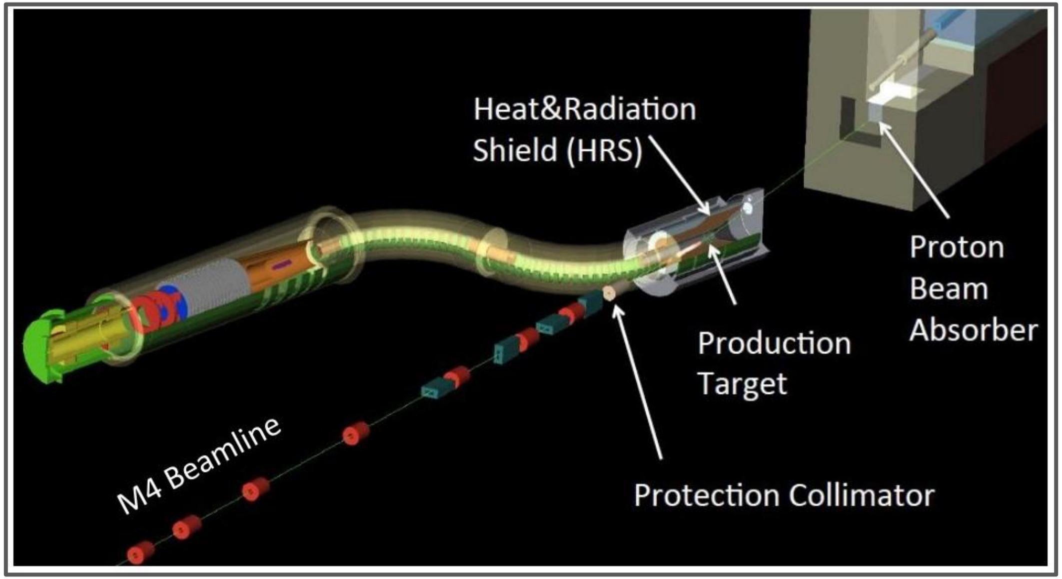
Supervisor
Kavin Ammigan
Co-supervisor
Zunping Liu

Fermilab

Proton Beam Dump

The Proton Beam Dump is a component made of **seven steel plates** surrounded by a steel baffle and **concrete** that absorbs the proton beam.

It needs a **Heat Removal System** where air is forced to circulate in order to keep the **concrete** temperature **under 95 °C**.

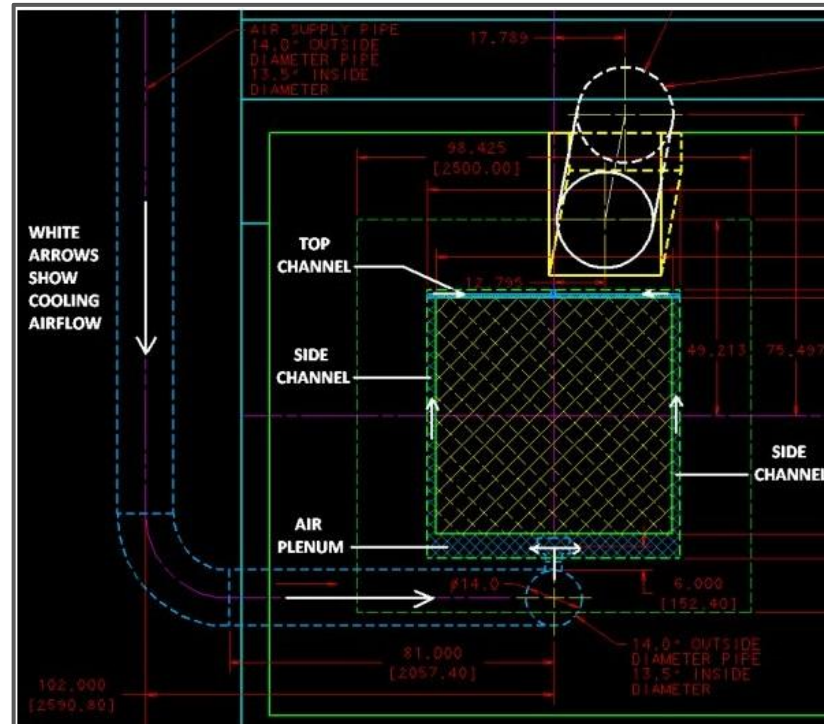


Picture from the Mu2e Technical Design Report, page 4-211

Task description

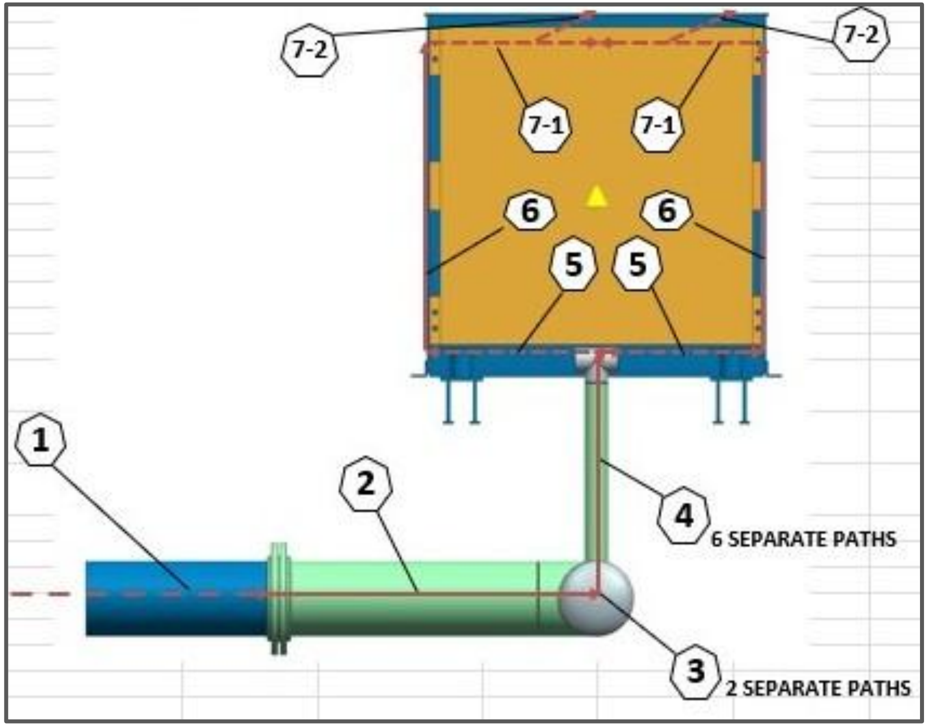
Verification of the **operative conditions** of the **Heat Removal System** of Mu2e Proton Beam Dump

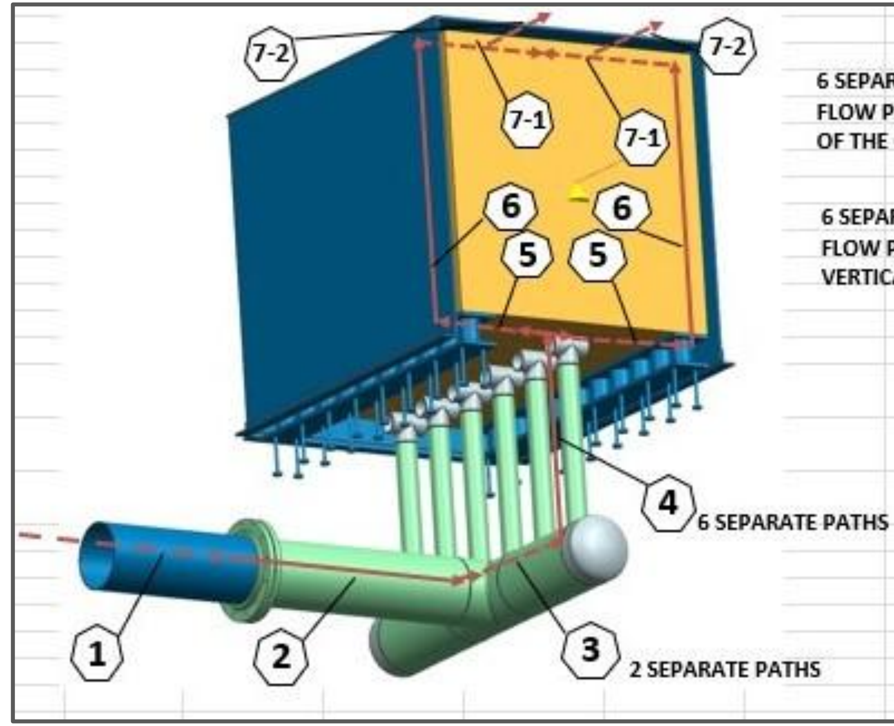
- 165 CFM vs 250 CFM
- Arbitrary power distribution generated by MARS code
    - Accident condition (6.7 kW)
    - Normal operation (1.3 kW)



Picture from the Mu2e Technical Design Report, page 4-246

Airflow path





From the building to the outlet

Pictures from the Excel file “Mu2e Airflow Calculations” by Andy Stefanik

Checking the results of the Excel file

Correction of **minor errors** in turbulent flow correlations

- P.K. Swamee and A.K. Jain formula
- Colebrook formula

Rewriting of the **sheet formulas** in clearer **VBA language**

Flow path number		1	2	3	4	5	6	7-1	7-2
		Building supply duct	Crossover line from building air duct to supply header	Supply header	Individual feed pipe	Horizontal gap under the core	Vertical gap between core and wall	Horizontal gap on top of the core 1 - lateral flow	Horizontal gap on top of the core 2 - longitudinal flow
Heat transfer coefficient - *Colburn Equation, +/- 30%*	W/(m^2-K)	4.1	4.1	2.3	7.0	0.9	2.9	3.0	14.2
Smooth tube Darcy friction factor	-	0.0265	0.0265	0.0319	0.0319	0.0508	0.0528	0.0556	0.0394
Nusselt Number - *Petukhov Equation, for liquids (10,000 <= Re <= 5,000,000*	-	48	48	30	30	11	11	10	18
Heat transfer coefficient - *Petukhov Equation, +/- 6%*	W/(m^2-K)	3.6	3.6	2.2	6.6	1.1	3.5	3.8	14.7
Use the smaller heat transfer coefficient	W/(m^2-K)	**3.6**	**3.6**	**2.2**	**6.6**	**0.9**	**2.9**	**3.0**	**14.2**
Tsurface - Taverage bulk air	C	0.000	0.000	0.000	0.000	32.049	99.902	105.545	0.000
Calculate the surface temperature (CHECK INPUT)	C	20.000	20.000	20.000	20.000	62.498	171.504	261.662	209.928
		OK	OK	OK	OK	OK	OK	OK	OK

Results with **165 CFM**: **very small heat transfer coefficients**

Flow path number		1	2	3	4	5	6	7-1	7-2
		Building supply duct	Crossover line from building air duct to supply header	Supply header	Individual feed pipe	Horizontal gap under the core	Vertical gap between core and wall	Horizontal gap on top of the core 1 - lateral flow	Horizontal gap on top of the core 2 - longitudinal flow
Smooth tube Darcy friction factor	-	0.0239	0.0239	0.0285	0.0285	0.0439	0.0451	0.0473	0.0345
Nusselt Number - *Petukhov Equation, for liquids (10,000 <= Re <= 5,000,000*	-	65	65	40	40	15	14	13	24
Heat transfer coefficient - *Petukhov Equation, +/- 6%*	W/(m^2-K)	4.9	4.9	3.0	8.9	1.4	4.4	4.7	19.2
Use the smaller heat transfer coefficient	W/(m^2-K)	**4.9**	**4.9**	**3.0**	**8.9**	**1.3**	**4.0**	**4.1**	**19.2**
Tsurface - Taverage bulk air	C	0.000	0.000	0.000	0.000	23.043	72.184	76.468	0.000
Calculate the surface temperature (CHECK INPUT)	C	20.000	20.000	20.000	20.000	53.339	139.853	220.341	193.004
		OK	OK	OK	OK	OK	OK	OK	OK

Results with **250 CFM**: still very **small heat transfer coefficients**

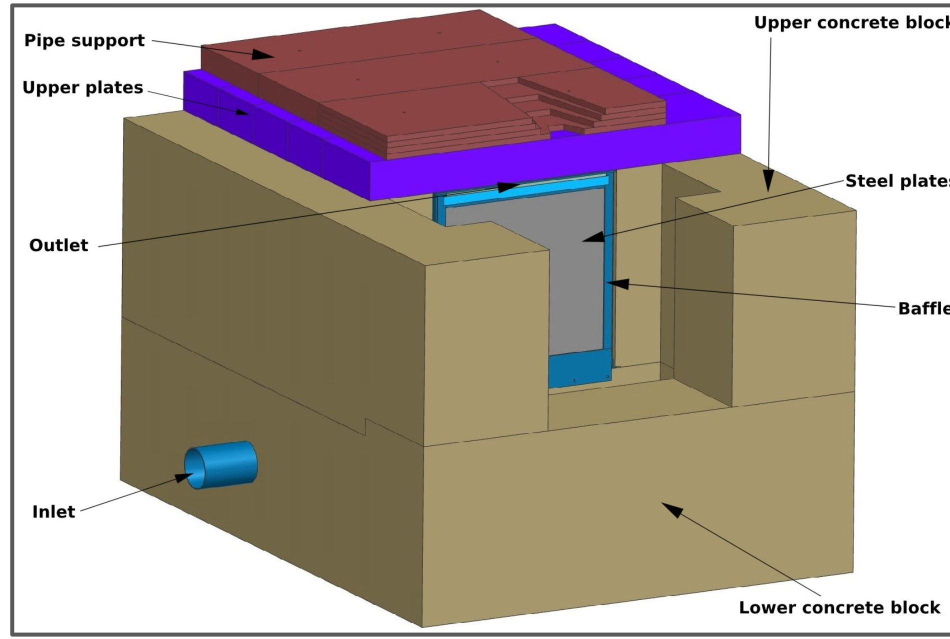
CAD Elaboration

Both solid and fluid components file CADs already available

Merging of the solid CAD and the fluid CAD in one assembly

Checking for **coincidence of the to-be-coupled surfaces**

Refinement of details (removal of holes, small imperfections) to facilitate the meshing and coupling process



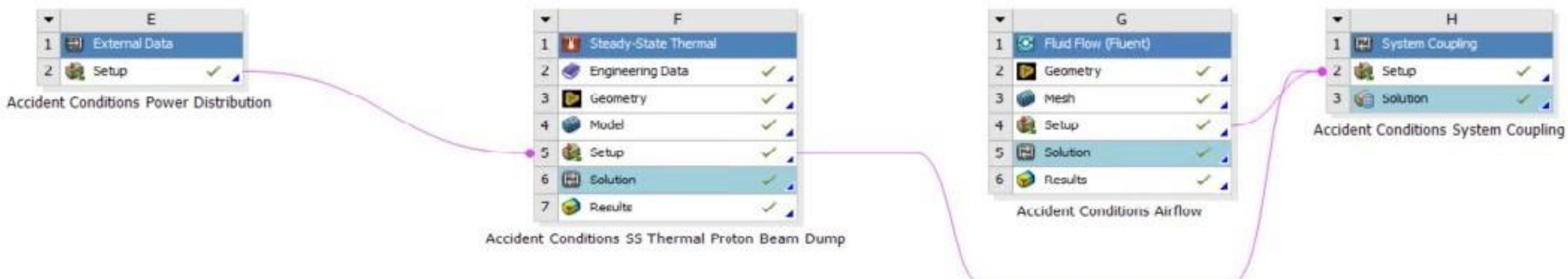
Picture from the Mu2e Technical Design Report, page 4-246

ANSYS Coupled Simulation

Necessity of a coupled simulation to get the **most accurate results**

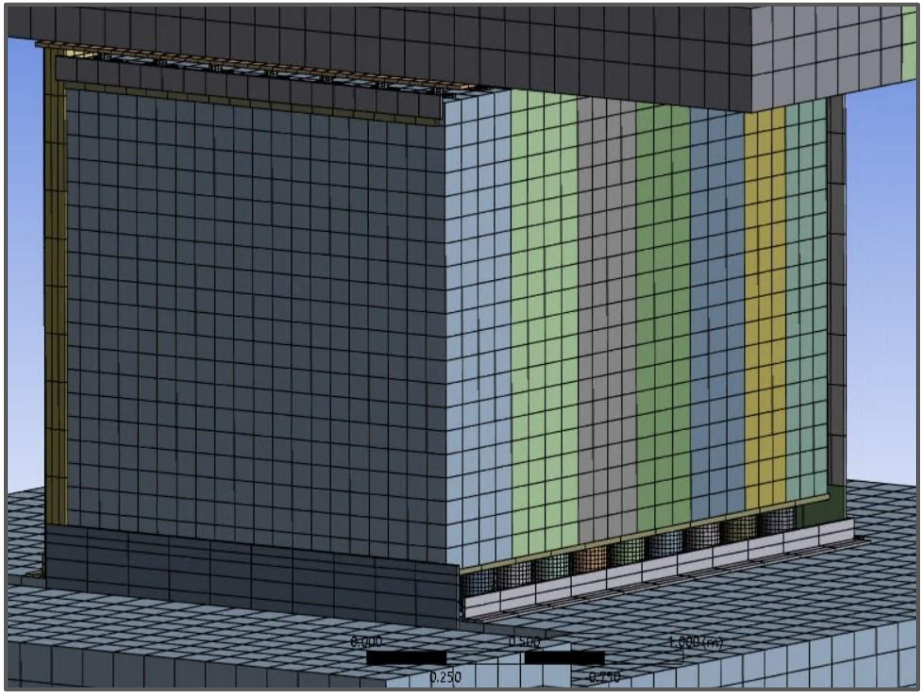
Decision of the **blocks** to put inside of the simulation

ANSYS offers the System Coupling block that couples **different phases**



Connections between the blocks in ANSYS Workbench

Solid meshing



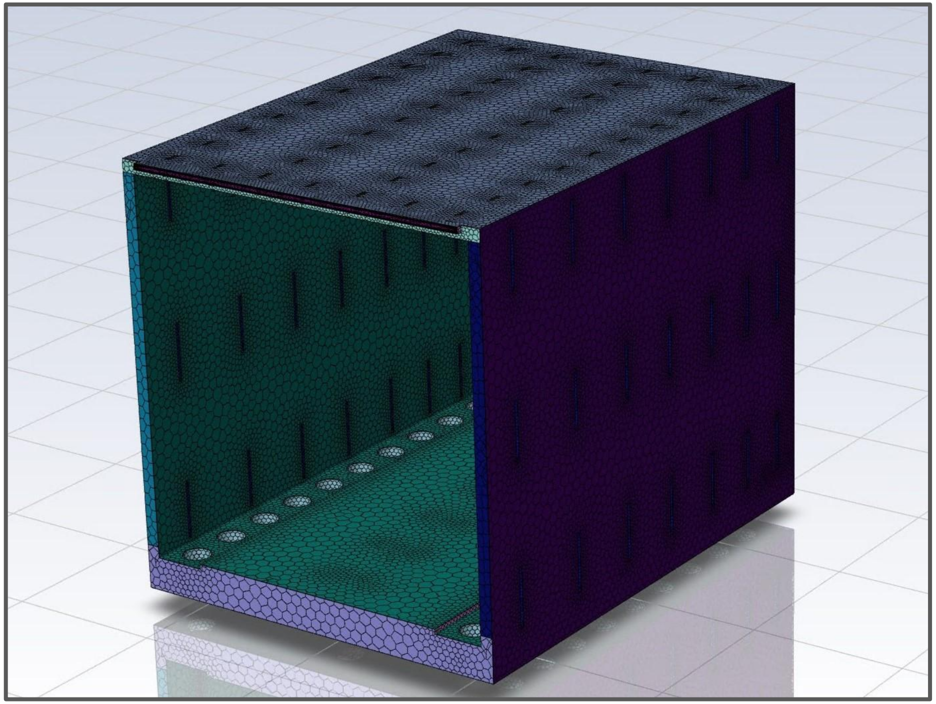
Solid meshing

Number of elements ~ **100’000**

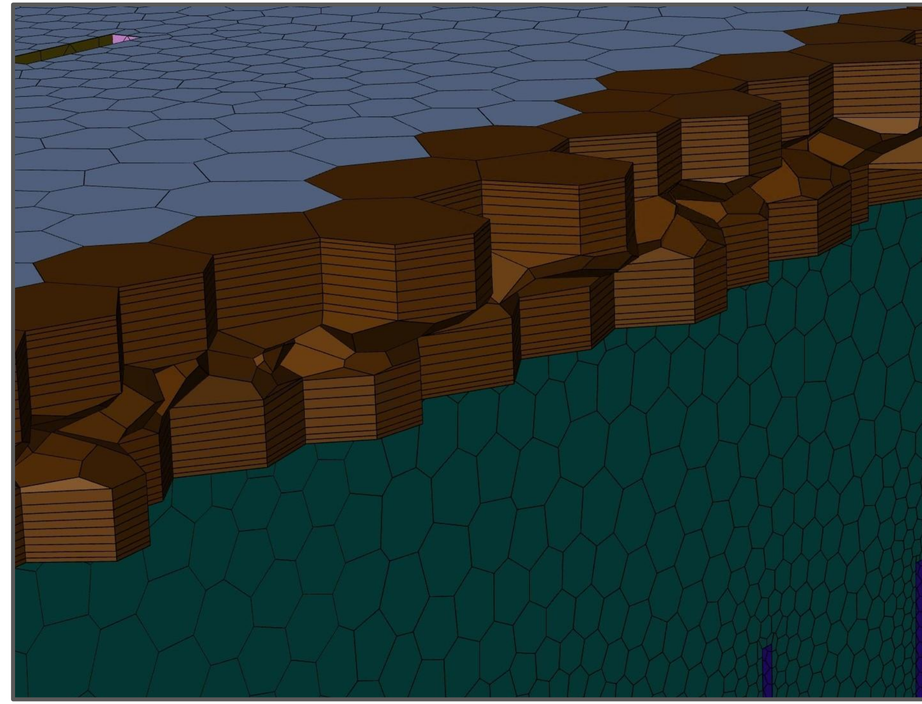
Quality Criterion	Warning Limit	Error (Failure) Limit	Worst
Max Aspect Ratio	Default (5)	Default (1000)	28.327
Min Element Quality	Default (0.05)	Default (5e-04)	0.038
Min Jacobian Ratio (Corner Nodes)	Default (0.05)	Default (0.025)	0.123
Min Jacobian Ratio (Gauss Points)	Default (0.05)	Default (0.025)	0.255
Max Element Edge Length	Default (3.716 m)	Default (7.432 m)	0.316 m
Max Corner Angle	Default (150 °)	Default (170 °)	166.64 °
Min Element Edge Length	Default (0.037 m)	Default (3.7e-03 m)	1.1e-03 m
Max Skewness	Default (0.9)	Default (0.999)	0.991
Min Tet Collapse	Default (0.1)	Default (1e-03)	0.146
Max Warping Angle	Default (20 °)	Default (30 °)	NA

Quality check

Fluid meshing



Prism layering



Boundary layer

Number of elements ~ **1’600’000**

Fermilab

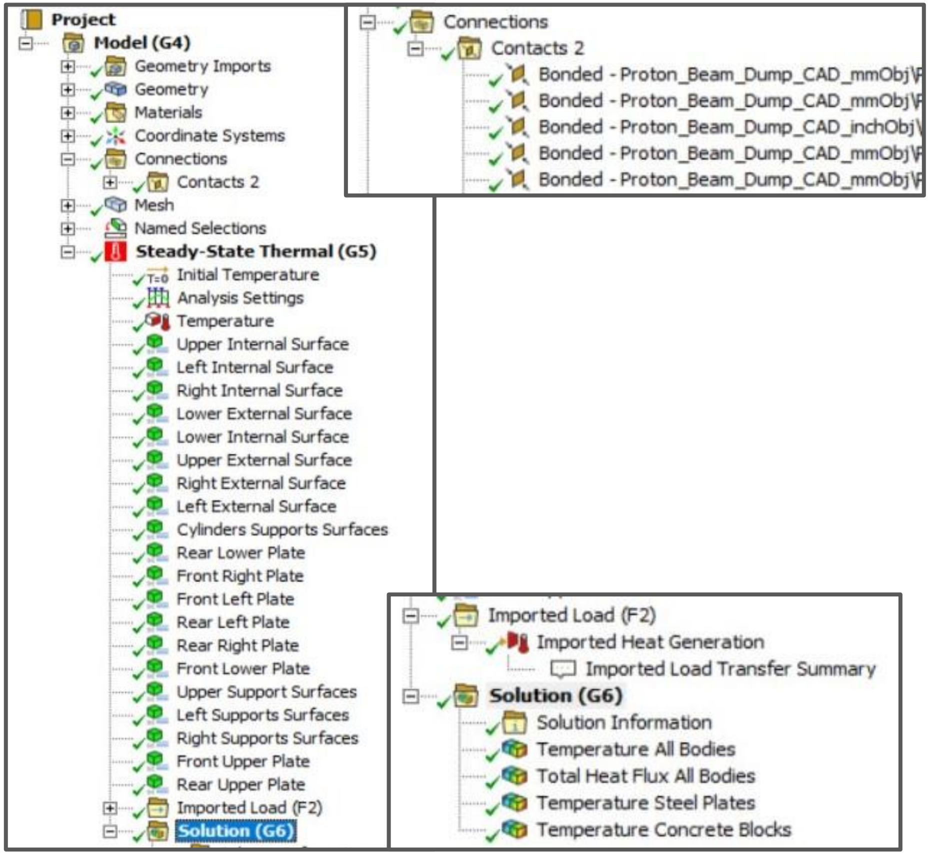
Boundary conditions - Solid

Imposed 15 C temperature on the bottom surface of the lower concrete block

Perfect insulation as default conservative assumption for all the external surfaces

Contact tool with imposed thermal conductance to simulate static air between baffle and concrete

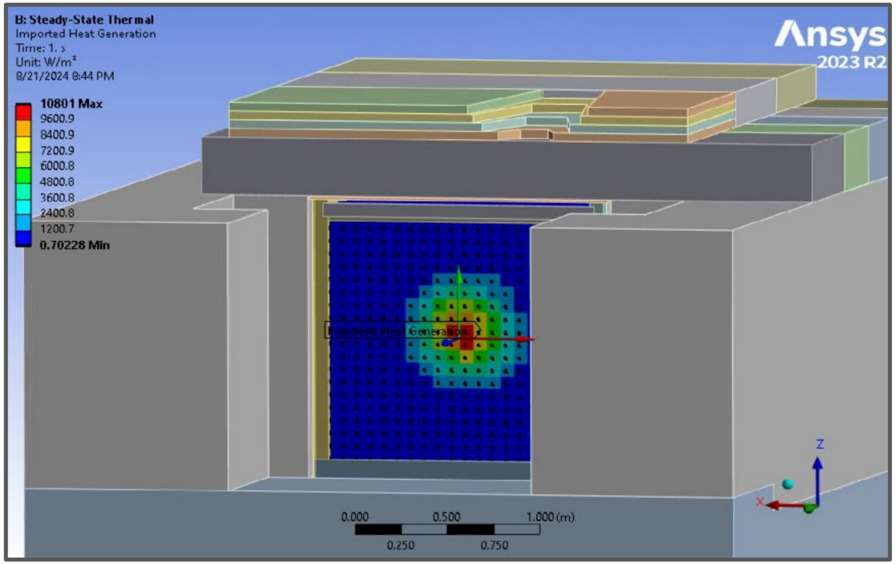
System Coupling Surfaces: 20 contact surfaces with the fluid where data is exchanged



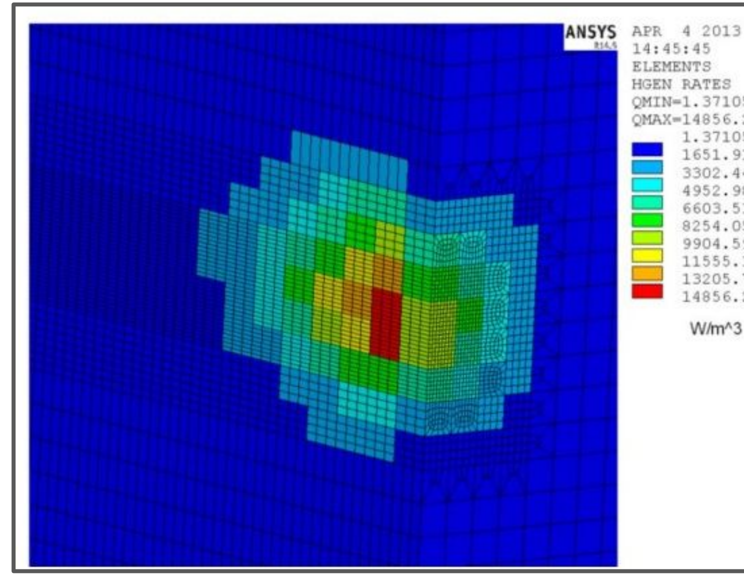
Fermilab

Boundary conditions - Normal distribution

Heat generation from MARS distribution with **1.325 kW of power deposited locally**



Mapped heat distribution (**obtained**)

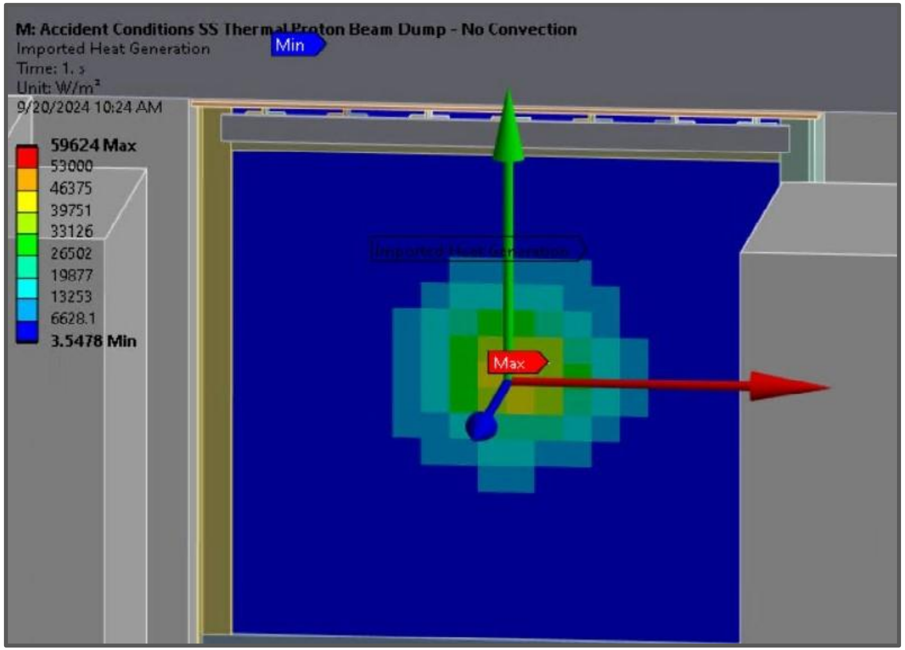


Picture from the Mu2e-doc-5048_The Proton Absorber for Normal Operating_Mu2e, page 2

Mapped heat distribution (**previous**)

Boundary conditions - Accident distribution

Heat generation from MARS distribution with **6.700 kW of power deposited locally**



Mapped heat distribution (**obtained**)

Mapped heat distribution (**previous**) not available

Boundary conditions - Fluid

Imposed **inlet velocity**

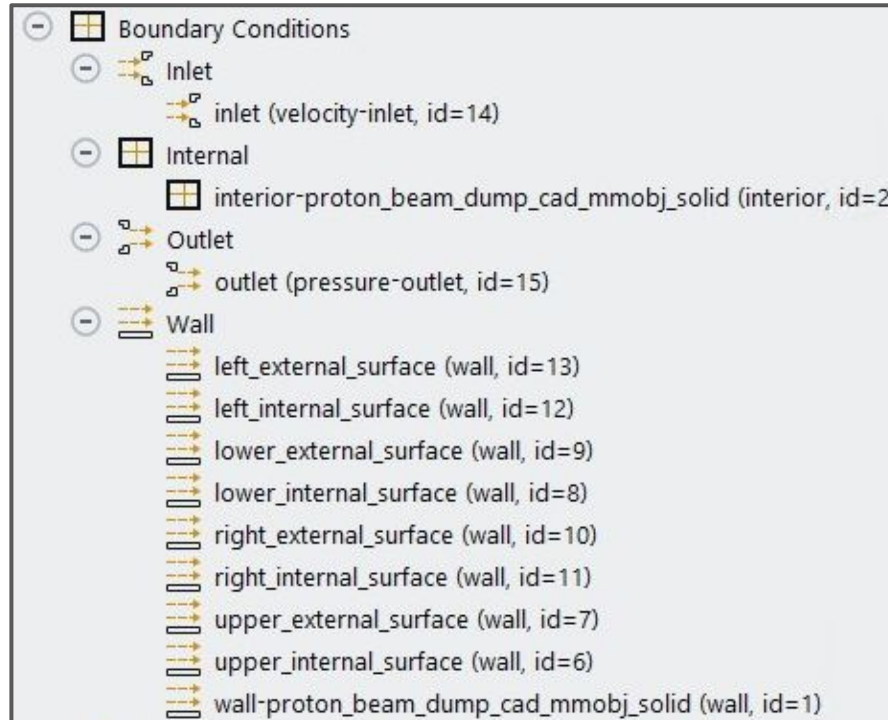
Imposed **outlet gauge pressure**

Fluid properties: **air**

Energy equation: **on**

Thermal boundary conditions: **via System Coupling surfaces**

Viscous model: **SST k-omega**



Fermilab

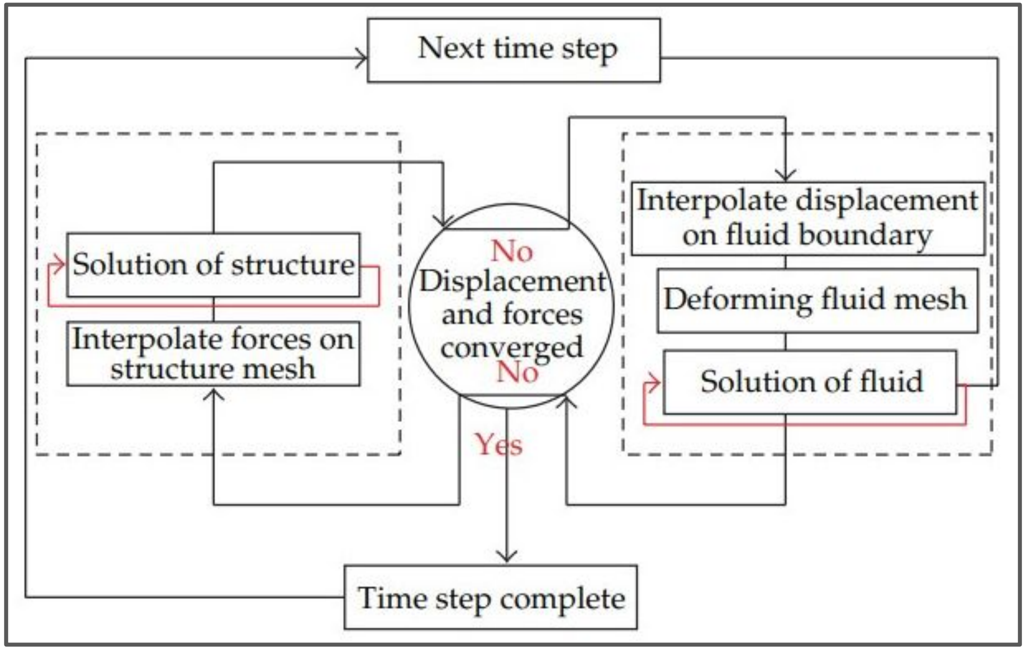
Two-way data transfer

Two-way data transfer

Data is initialized in the uncoupled simulation and **transferred on the coupled interfaces**

More coupling iterations to get the **converged and common value**

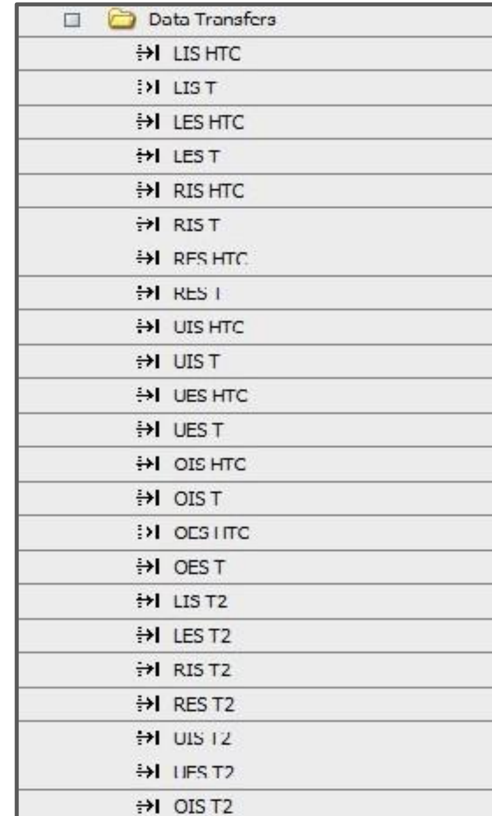
In a **steady-state** simulation, there is only one time step (external loop)



Coupling conditions

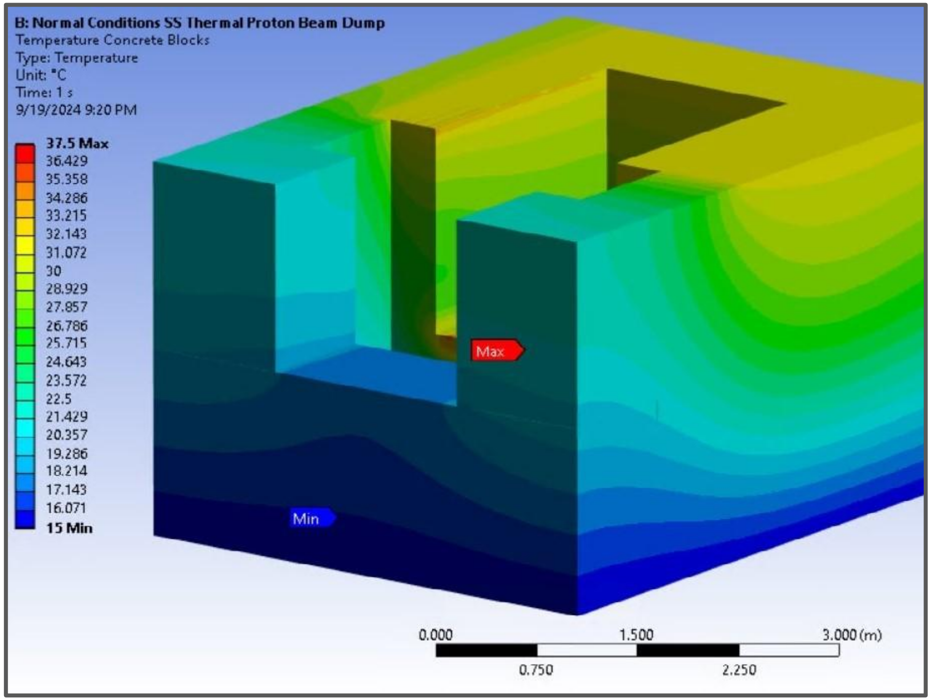
To make the coupled convection work, for each surface three transfers are needed :

- **Heat Transfer Coefficient** from Fluent simulation to **Convection Coefficient** of Thermal SS simulation
- **Near Wall Temperature** from Fluent simulation to **Convection Reference Temperature** of Thermal SS simulation
- **Temperature** from Thermal SS simulation to **Temperature** Fluent

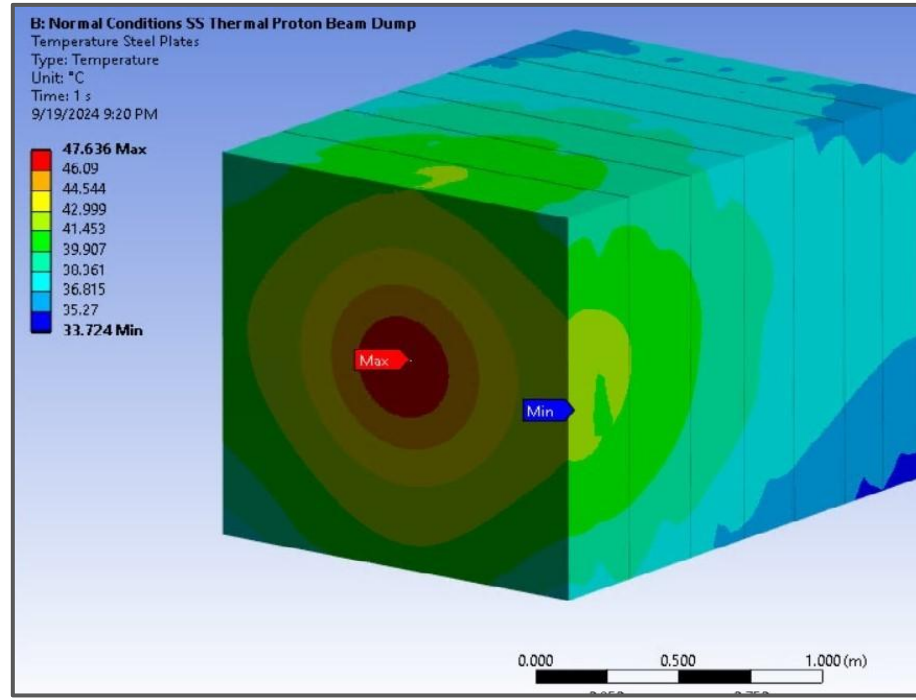


Fermilab

Normal heat distribution - 165 CFM

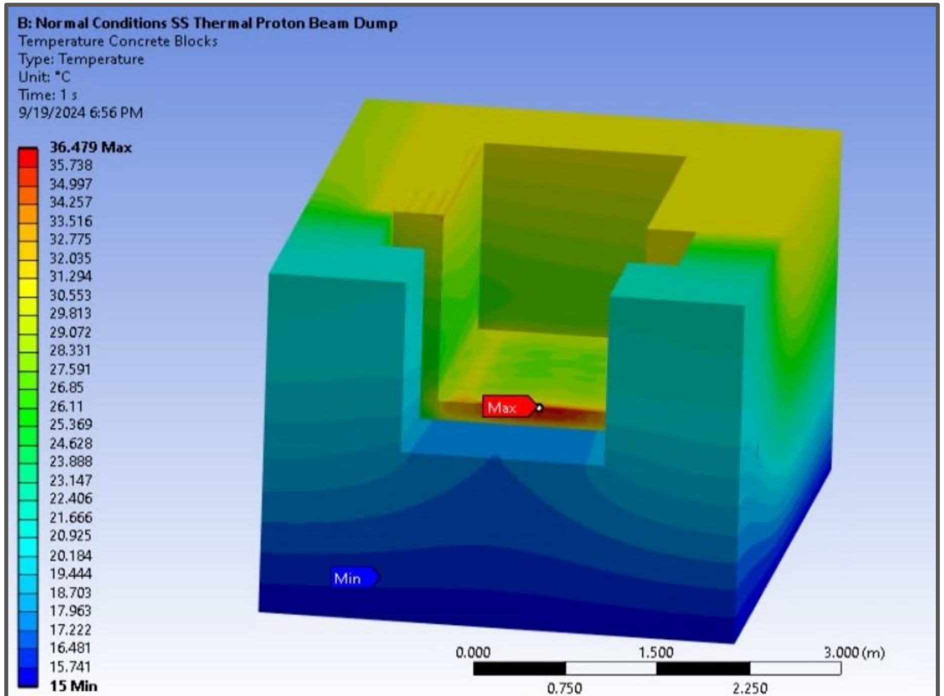


Concrete temperatures

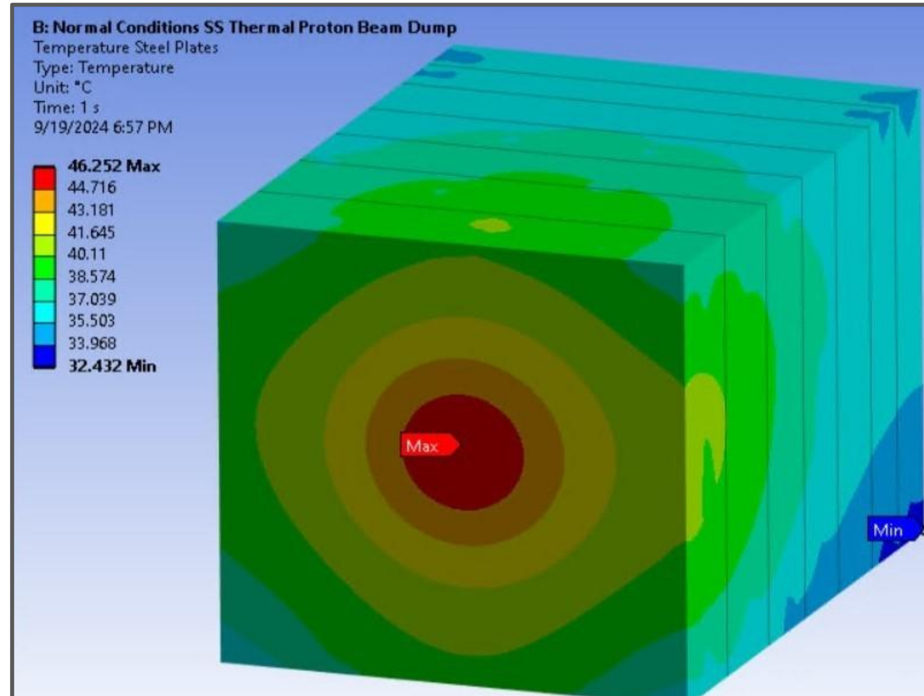


Steel temperatures

Normal heat distribution - 250 CFM

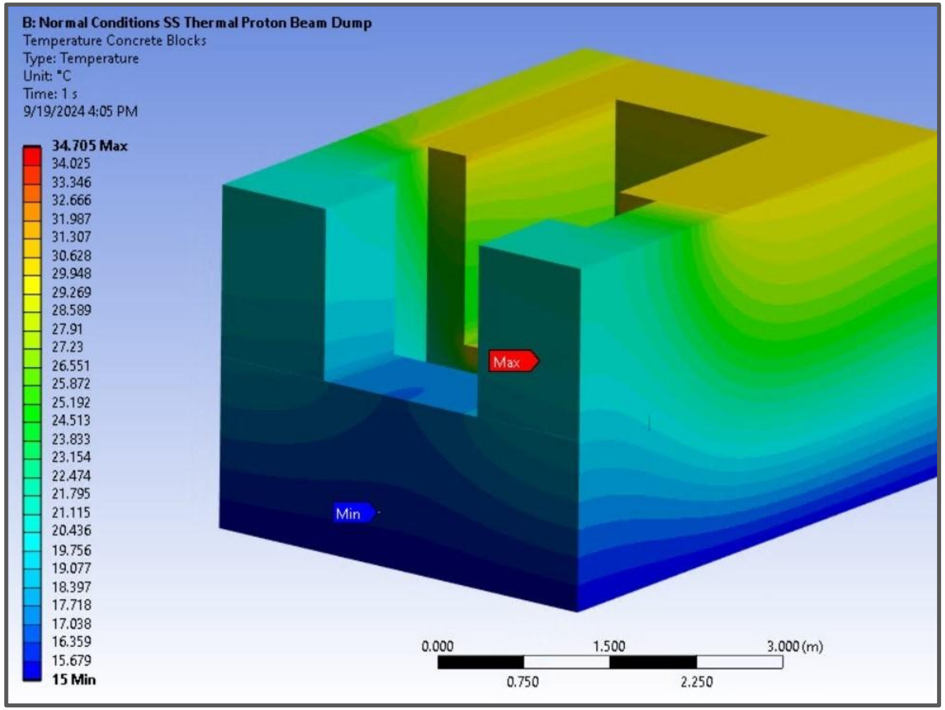


Concrete temperatures

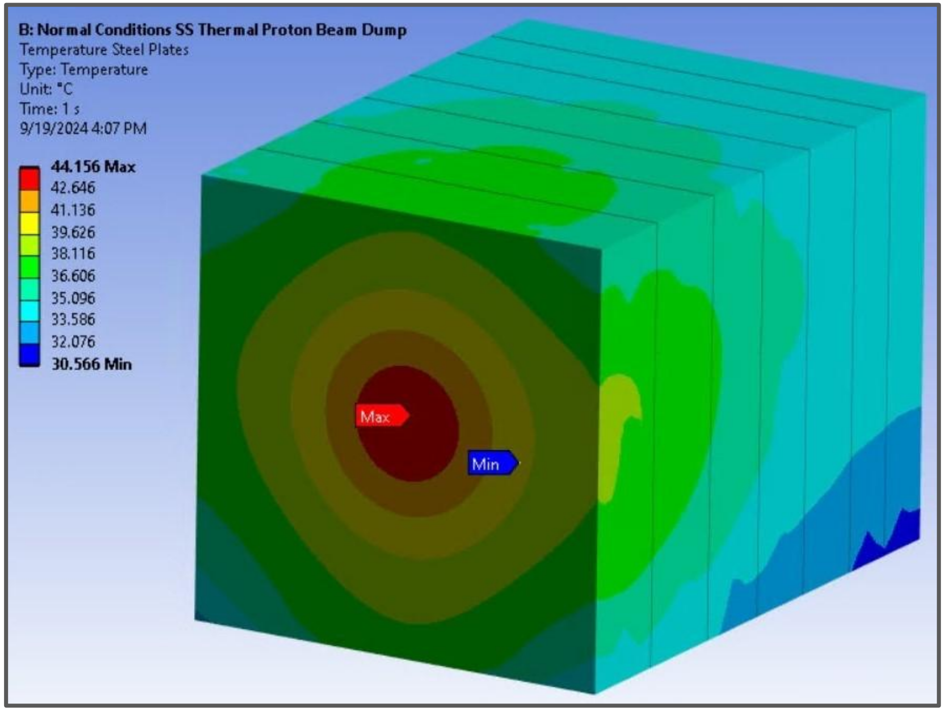


Steel temperatures

Normal heat distribution - 400 CFM

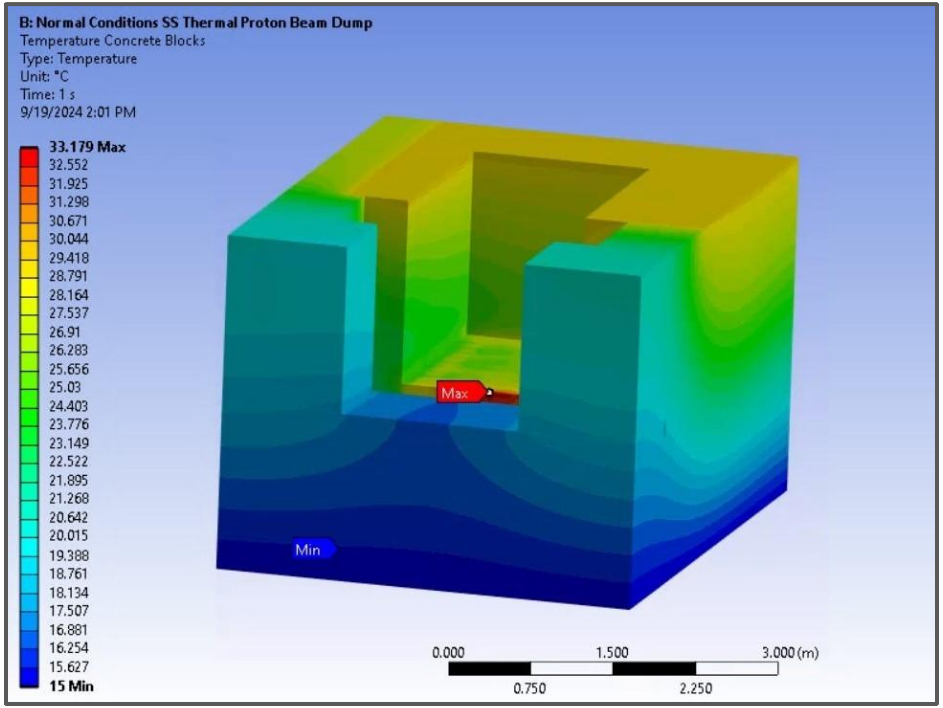


Concrete temperatures

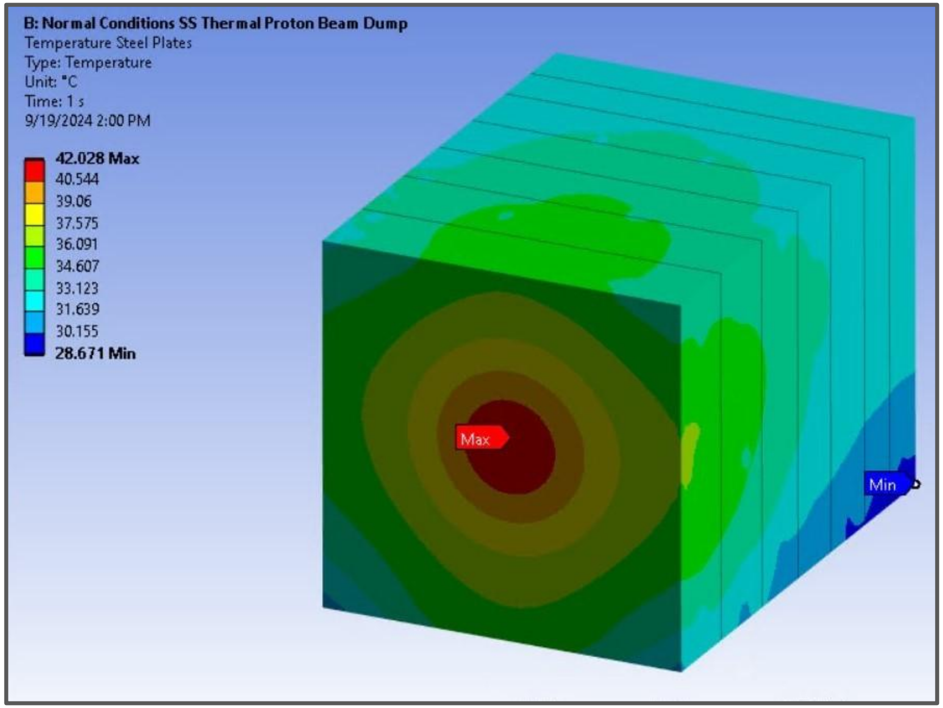


Steel temperatures

Normal heat distribution - 600 CFM



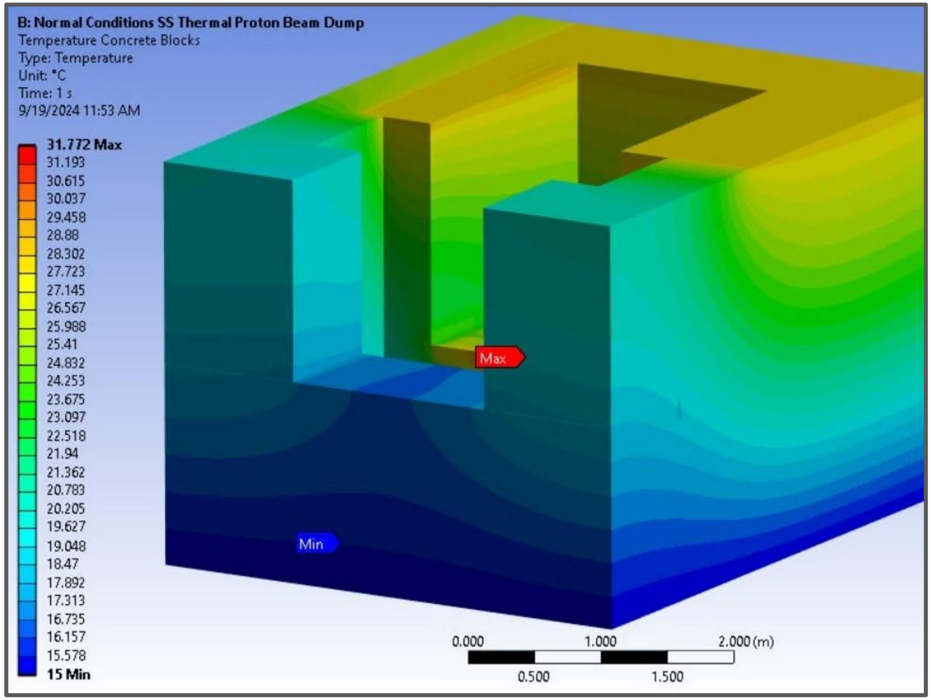
Concrete temperatures



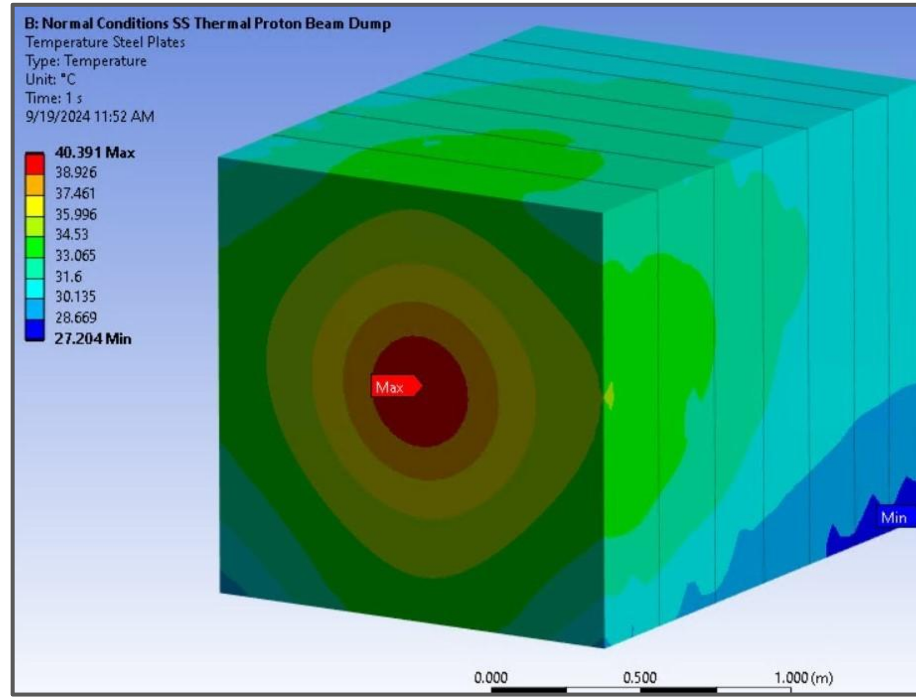
Steel temperatures

Fermilab

Normal heat distribution - 800 CFM

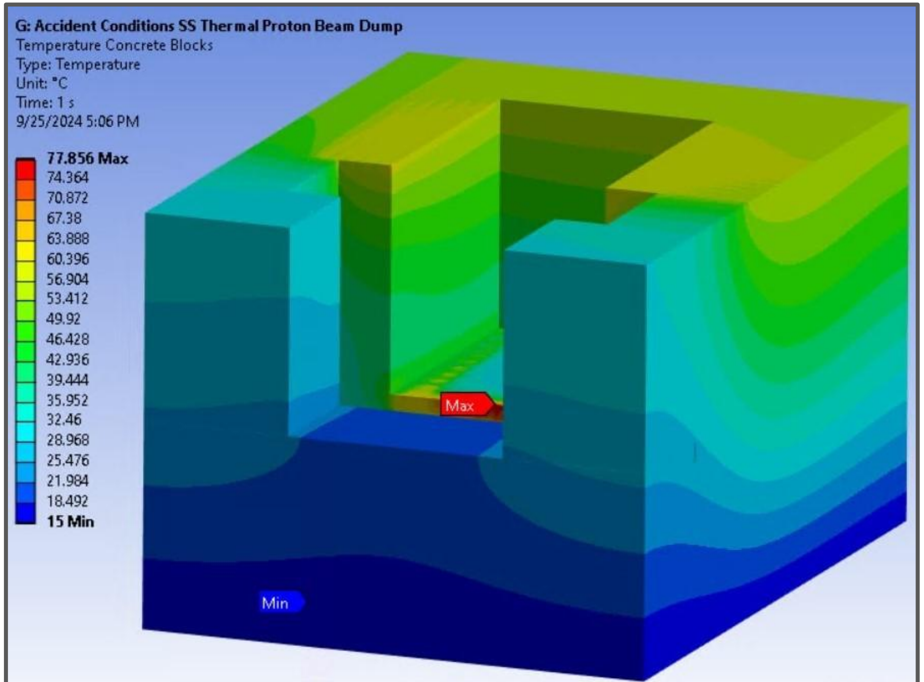


Concrete temperatures

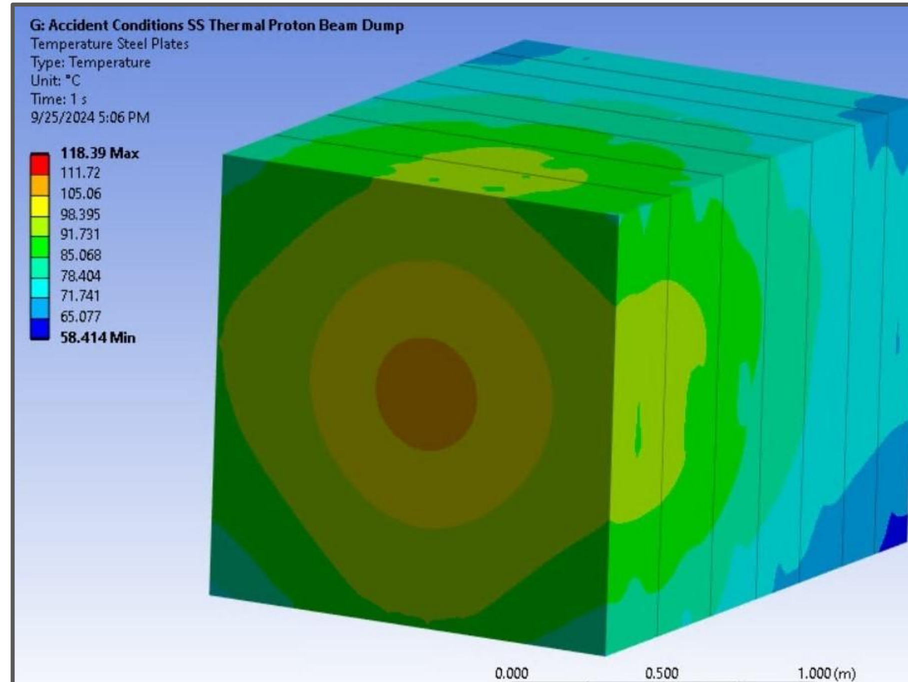


Steel temperatures

Accident heat distribution - 165 CFM



Concrete temperatures



Steel temperatures

Global mass and energy balance equations

Heat distribution	Airflow [CFM]	Inlet flow rate [kg]	Outlet flow rate [kg]
Normal	165	0.092	-0.092
Normal	250	0.14	-0.14
Normal	400	0.22	-0.22
Normal	600	0.33	-0.33
Normal	800*	0.46	-0.46
Accident	165	0.092	-0.092

Table 6.2: Mass flow rates

$\Delta T_{\text{simulation}}$ should be equal to $\Delta T_{\text{energy balance}} = \dfrac{Q - Q_{ext}}{c_p \dot{m}}$

Heat deposited	Airflow [CFM]	$\dot{m}$ [$\frac{\text{kg}}{\text{s}}$]	Q_{ext}[W]	$\Delta T_{\text{simulation}}$[°C]	$\Delta T_{\text{energy balance}}$[°C]
1307	165	0.0954	127.7	11.8	12.9
1307	250	0.145	119.8	10.3	8.2
1307	400	0.231	108.3	8.59	5.2
1307	600	0.347	97.4	7.1	3.5
1307	800*	0.463	44.9	1.9	2.8
6225	165*	0.0920	255.3	38.5	64.5

Table 6.3: Global heat check

Comparison with the Excel

Region	Value $[W/m^2K]$
Horizontal gap under the core	2.0
Vertical gap between core and wall	4.5
Horizontal gap on top of the core - lateral flow	4.2
Horizontal gap on top of the core - vertical flow	11.6

Table 5.29: HTC values from the Excel file

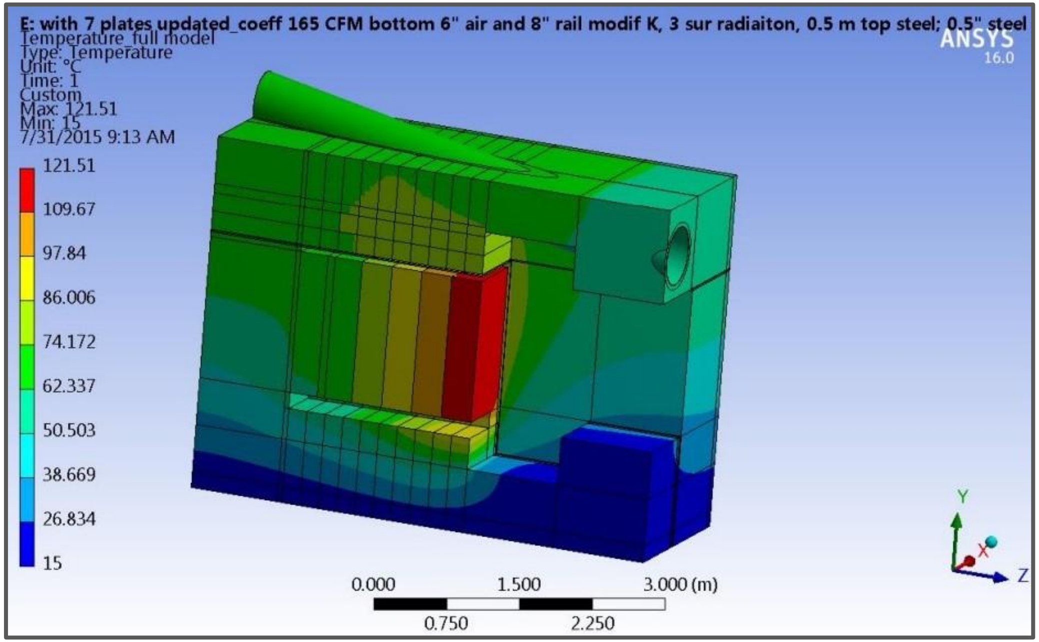
Excel HTC

Region	Value $[W/m^2K]$
Cylinders Supports Surfaces	12.2
Lower External Surface	9.9
Lower Internal Surface	9.7
Front Lower Plate	10.1
Left External Surface	11.3
Right External Surface	11.4
Left Internal Surface	10.9
Right Internal Surface	11.1
Rear Left Plate	9.7
Rear Right Plate	9.1
Front Left Plate	14.6
Front Right Plate	14.7
Left Supports Surfaces	11.8
Right Supports Surfaces	11.7
Upper Supports Surfaces	12.6
Upper Internal Surface	12.7
Upper External Surface	12.3
Front Upper Plate	19.5
Rear Upper Plate	8.7
Rear Lower Plate	8.9

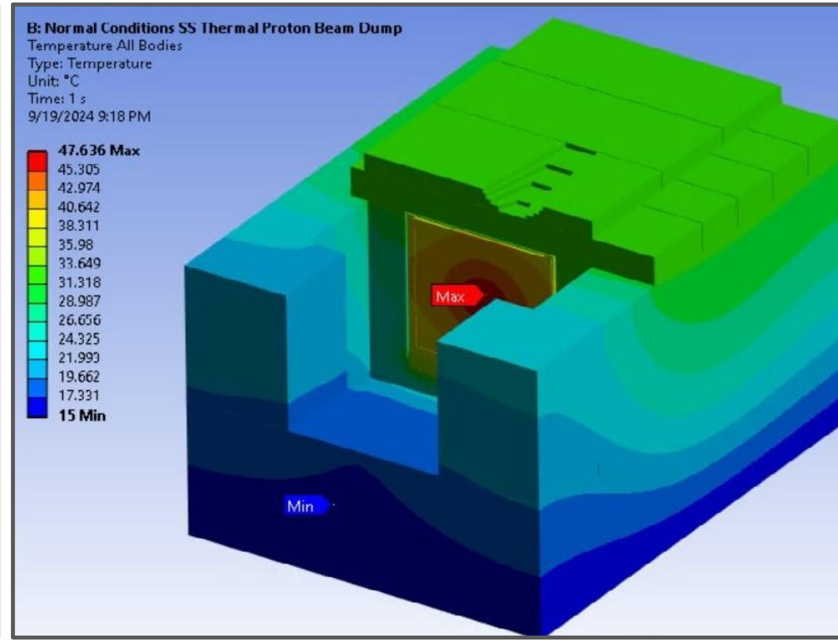
Table 5.30: Wall adj. HTC values from the ANSYS coupled simulation

Simulations HTC

Comparison with the ANSYS Thermal



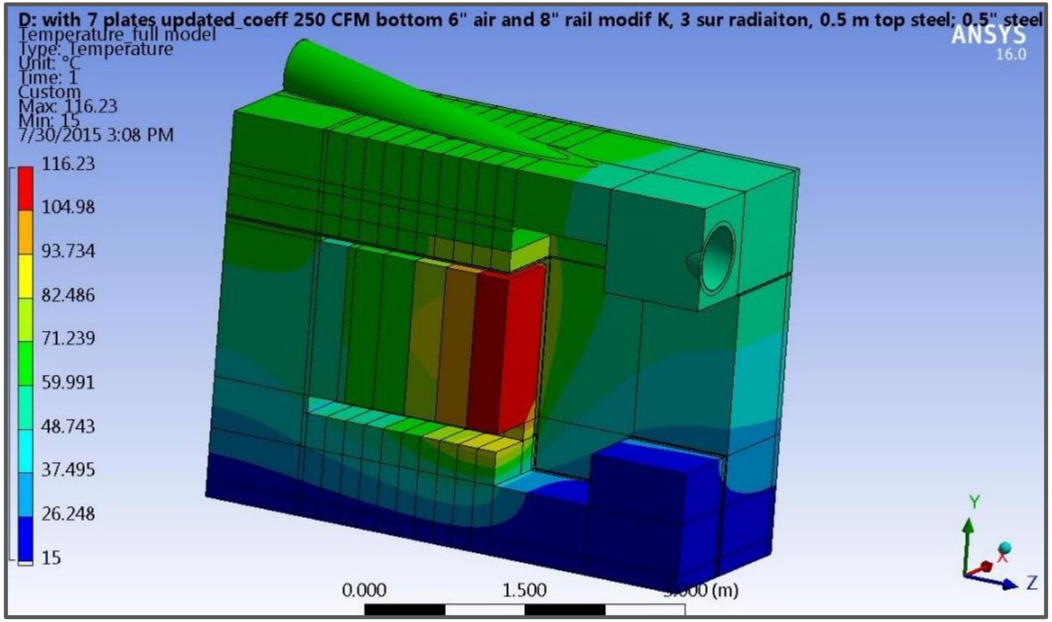
ANSYS Thermal temperature
Normal h. d. - 165 CFM



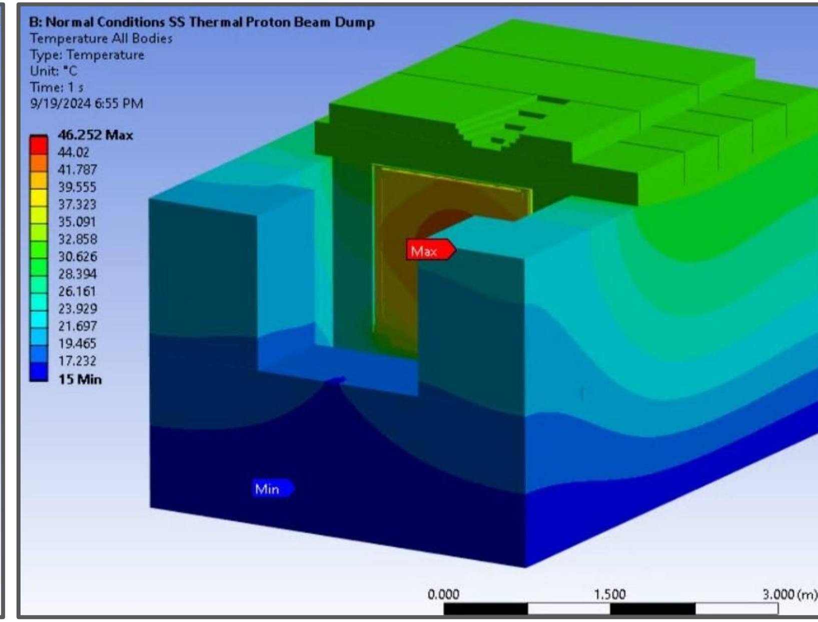
ANSYS Coupled temperature
Normal h. d. - 165 CFM

Fermilab

Comparison with the ANSYS Thermal



ANSYS Thermal temperature
Normal h. d. - 250 CFM



ANSYS Coupled temperature
Normal h. d. - 250 CFM

Fermilab

Contributes to the project

- The already available heat transfer coefficient values are checked to be **conservative values**
- The temperatures reached in the coupled simulations are **lower than those estimated by the correlations**
- The peak of the temperature distribution in the concrete block appears in the **front lower plate region**
- The coupled simulation provides a **more accurate estimation of the pressure drops** (useful to calculate the necessary fan power)
- The **practical evaluation of static air and radiative contributions** can be implemented from my calculations
- A **general framework** was created to set up another beam dump simulation for possible future experiments
- A **complete and benchmarked software (ANSYS)** usually well-known by structural engineers has been used
- **Easier and faster change of parameters** as the MARS heat distribution, the volumetric flow, the geometry
- **Twenty different coupled surfaces** where it is possible to evaluate the film coefficients, temperatures and coupling conditions

Why the simulations results should be accurate

- The **solid component converges** and does not present residual problems
- The **fluid component residuals are low**. The energy residual is below 2e-3 and the omega and k residuals are below 5e-3 for all simulations
- The **coupled simulations residuals** are below 1e-2 and **below the default values of convergence**
- The **real heat distributions have been substituted** with uniform heat distributions keeping the power constant **getting very close results**
- **Individual quantities** as velocity at the outlet **were observed to converge** within 200 iterations without oscillatory behaviour
- Mass flow rates at both the inlet and the outlet were checked for consistency. **No significative mass generation or mass destruction has been observed** inside of the volume, according to the global continuity balance
- **Reducing the air volumetric flow makes the temperature** of the system **go up**, as it should
- Turbulent quantities as **turbulence intensity and turbulent viscosity ratio were checked to be correct**
- The increase in temperature of air is **approximately consistent with the global energy balance of the system**
- The number of elements of the meshes is quite high. **Solid elements: 100’000. Fluid elements: 1’600’000.**
- Uncoupled simulations were run to check the worst case scenarios
- About **97% of the surface of the fluid component** has been considered as inlet, outlet or coupled (the other 3% has the default perfectly insulated behaviour)
- The **viscous model adopted has been checked**. The flow is turbulent, having a Reynolds number > 2000
- The y+ values have been checked. **Every surface has y+ values under 5**, as requested by the SST k-w model
- The **correct coupling of the surfaces has been checked**. Values up to 100% of the coupling were reached

Why the simulations results could not be accurate

- It is the first time I do a coupled simulation. Maybe some data transfers **are not set up correctly**
- Air properties as specific heat, density and dynamic viscosity **have been considered not temperature dependent**
- Outlet conditions could be set up in a **more realistic way**. Far-field conditions could be applied to a volume external to the outlet
- Air could be better simulated on the faces of the baffle by using **more accurate static air models**
- The **heat distribution is not mapped perfectly**, even though the error on the total deposited power is less than 1%
- **Fluent residuals are still not too low** (due to mesh or geometry probably)
- **Radiation effects are not considered** in both solid and fluid components
- The global energy transfer **is still not quite exact**. There appears to be an unwanted source/sink of heat (convergence?)
- **Big differences of the volumetric flow** give only **small differences in temperature**
- **The HTC are higher** than what estimated by the classical correlations

Future improvements and corrections

Need of a transient simulation to know the developing of the situation in time (fluid SS and manually exchange data to thermal transient)

Mesh sensitivity study to check what is the role of the mesh (both solid and fluid component). The size of the geometry is big and the actual number of the elements could not be sufficient to describe what's happening

Check the limit of **very high flow** (should tend to bring the surface temperature of the block to 20 degrees celsius) and the limit of **very low flow** (should tend to the no-convection simulation). Note: possible use of laminar model in the last case

Coupling iterations sensitivity study: the total heat transmitted to the fluid is not the total heat flowing through the coupled surfaces of the solid. This means that **the heat flux and temperature** values of the coupled simulations **are not converged**.