



CERN Summer Student Program - Project Report

CFX Analysis for LAGUNA

Supervisor: Francisco Sanchez Galan

04/09/2014

Abstract

The purpose of the LAGUNA project is to unlock the secret of neutrinos which is one of the key elements of the universe. In order to achieve that, two underground facilities is required to build. One of these facilities will be in CERN and the other will be in Finland. The mission of the CERN is to generate neutrinos in this underground facility. So that, detectors in Finland will have the neutrinos to make the measurements. To generate the neutrinos protons at high speed are going to be needed. These protons will make collisions with a graphite rod (called target) in this facility. When the protons collide with the target, lots of subatomic particles will born. Powerful magnets will give direction to these particle towards to a decay tunnel where the particles turn into neutrinos and start their journey to detectors. Some particles, however, won't be affected from the magnet. They will interact with the walls of decay tunnel and target chamber. This will cause the heat generation inside the walls.

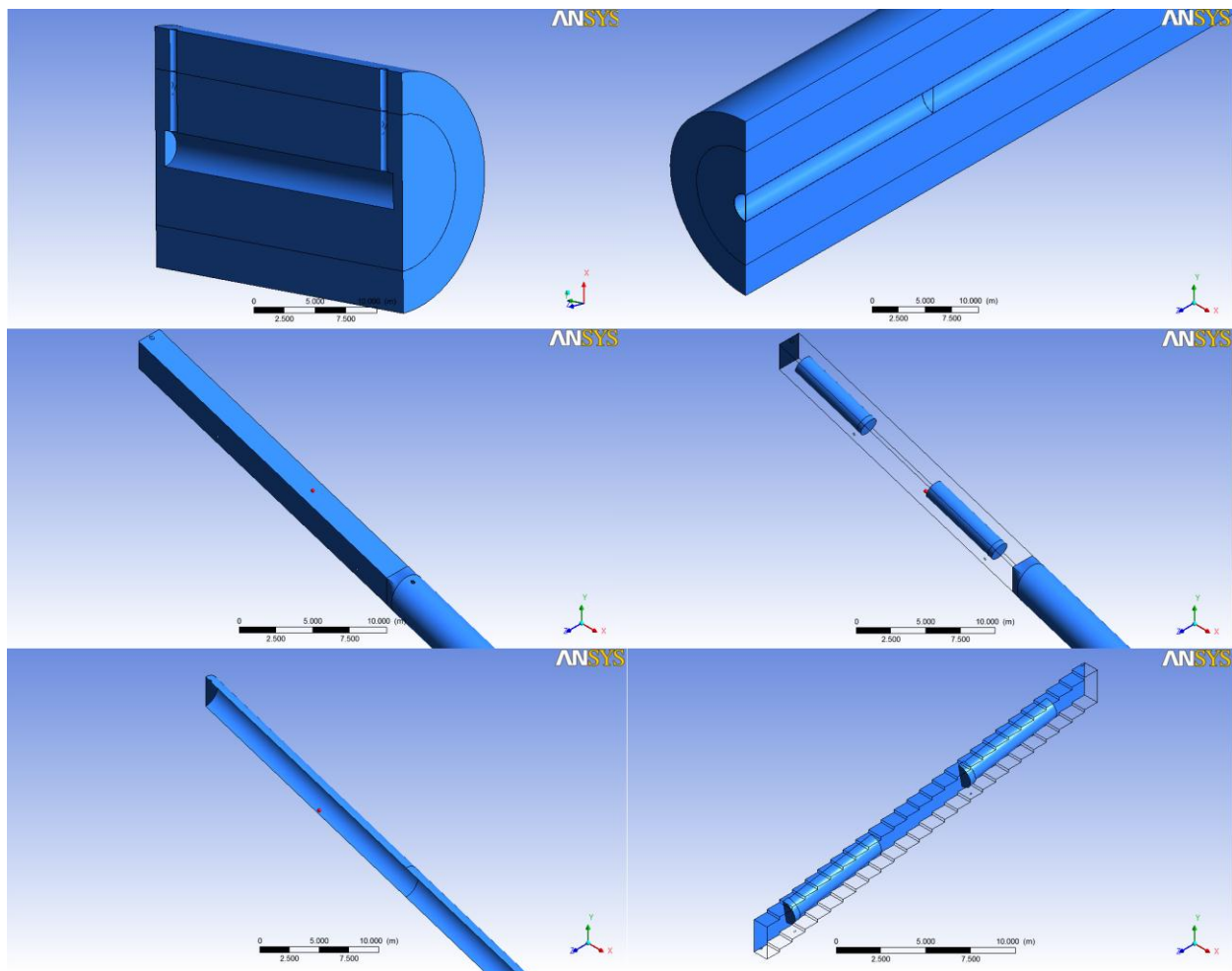
My duty in this project is to help to my supervisor to determine and analyze the required mechanism to remove heat from this structure by using Ansys CFX module (a finite element method interface for computational fluid dynamics).

Learning CFX

When I started to this project, I was not familiar with the CFX. I had generally used FLUENT module before coming to CERN. After many tutorial sheets, videos, tries and thanks to helps of my supervisor, I have finally figure it out how it works. It took some time for me but in the end it manage to handle. Actually the most of this time is consumed by modelling and iteration parts. CFX would be the minor part in this whole learning process.

CAD (Computer Aided Design) Modelling

After figuring out the CFX's logic, I have started my main purpose. I have started to design required domains in accordance to my supervisor's instructions using the Solidworks and Geometry Module of Ansys. I had to go with two different CAD interface because each one has its own upsides and downsides. While it is much easier to prepare the basic shape of the domain in Solidworks, it is easier to give details with Ansys Geometry module. Although I had only one coolant which is helium in the system, I had to prepare many different models. It was mostly because of the size of the structure. Since I did not have the enough processing power that can handle with the whole model of the structure, I had to divide it in to smaller parts in order to gain some time. I also modelled different kinds of versions of almost each divided parts for different scenarios and different methods. In the end I had plenty of CAD models to solve in CFX.

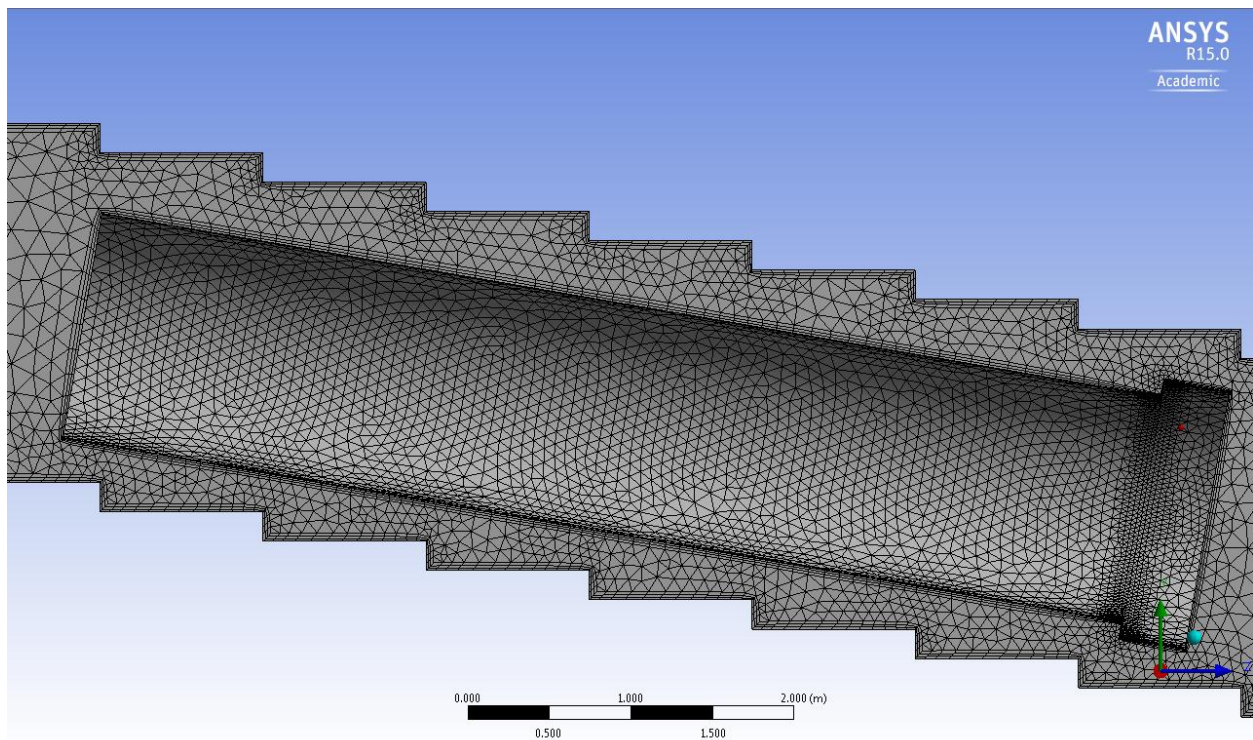


Some examples for the designs

Meshing

In order to prepare to CAD model for CFX, it need to be meshed. Meshing process is also a challenging and one of the most important stage for finite element method. It does not only need to have enough resolution to get correct results but also it has to be time efficient during iterations. In exceptional cases I have generally worked with a mesh number raging between 250 000 and 1 million. In different kind of domains, I have used different types of meshing methods. I carefully worked on boundaries since it is the most important part of the model. Because all of the interactions occurs on boundaries their resolution and quality drastically effects the results.

At the beginning I have used basic meshing methods to get the result fast. I knew that the result were not going to be accurate but they could give a feeling whether the boundary conditions set correctly.



Meshed Target Chamber

Setting the Boundary Conditions in CFX and Solutions

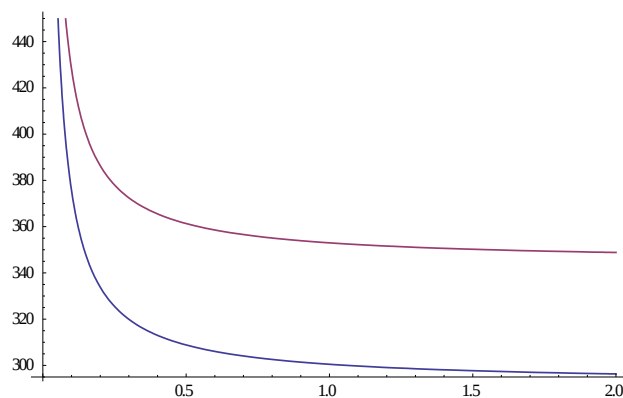
This part is just defining the conditions of boundaries but it requires a lot more caution. For example, setting the length 251m instead of 250m or using 3 layer inflation instead of 5 layer would affect the result but it wouldn't cause significant difference. Setting boundary conditions wrongly, however, would absolutely give wrong results, at least the results that we are not looking for.

At the beginning I needed material properties of helium, concrete and rock which are not defined in the library of CFX. After finding the material properties, I have defined the boundary conditions according to data that my supervisor gave me. Then, I have left it to computer to solve the model. The duration to solve the model was raging between 1 hours and 80 hours, depending on the quality of mesh and complexness of the model.

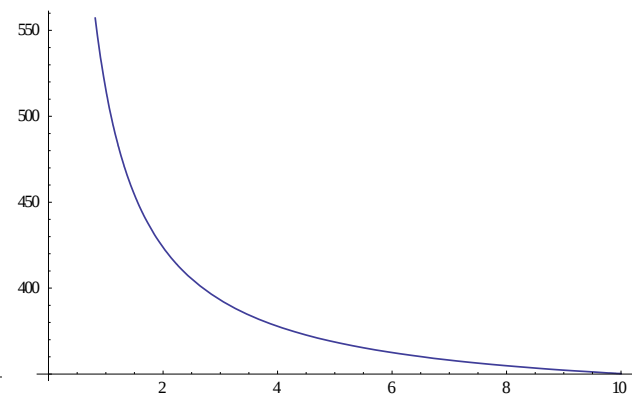
Different Scenarios and Theoretical Solutions

While the computer solving the problem, meanwhile I was preparing different models to try my own ideas on the solutions. I have discussed with my supervisor for eligible methods to understand how much options I have. Since we don't have too much options, I have started to play with flow regime. I knew that turbulence could increase the film coefficient so I have designed models that will increase the turbulence and different inlets from different points and angles.

At the same time I have write some scripts using Mathematica to calculate the heat transfer with theoretical methods (In other words hand calculations). I drew some figures that shows Temp. Vs Position, Film Coefficient vs Temp. Etc. I had to do these calculations to verify the CFX results. Although I made these calculation with assumptions and CFX made them consisting every effect such as gravity or turbulence, the gap between two results should not be too big. I also used FLUENT for one case again verify the results of CFX.



Temp Vs Mass Flow Rate – Red Surface Temp – Blue Helium Temp in [K] for $h=3.5$ -- for 43.3kw heat flow

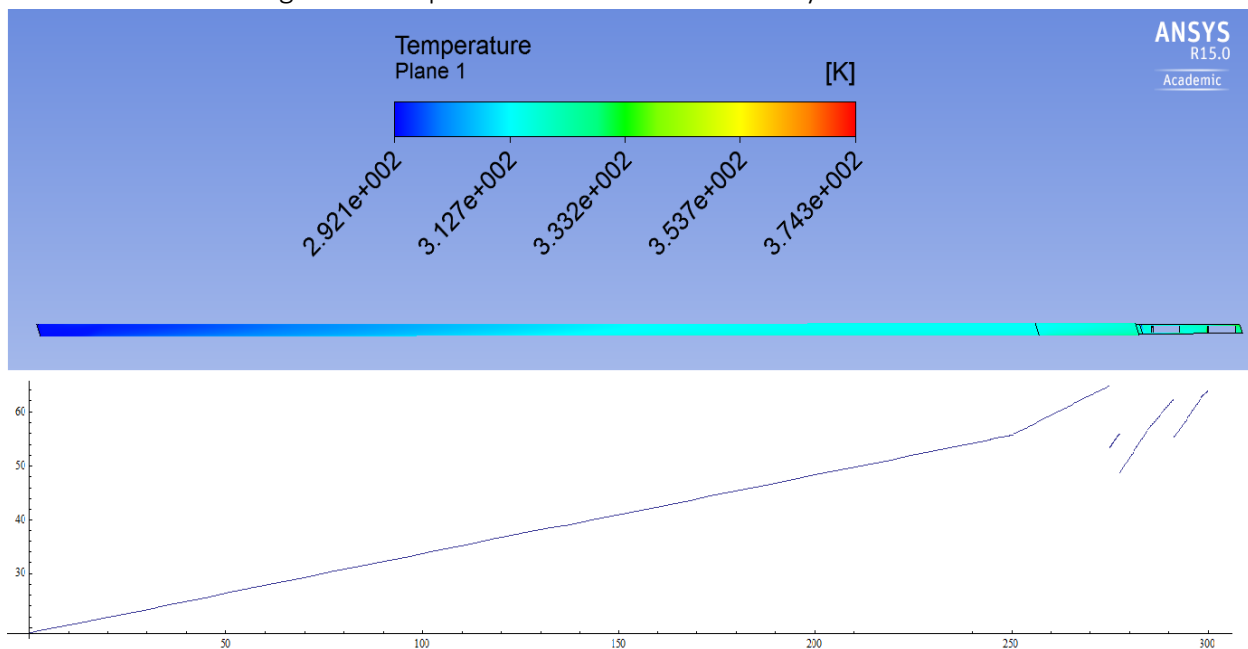


Surface Temp vs h -- mass flow rate=0.21kg/s --for 43.3kw heat flow

Results

The results that I obtained were quite satisfying. I have examined the results with my supervisor and discussed on how we could improve the efficiency of the cooling system. With some minor and major changes in CAD designs and boundary conditions I have prepared different CFX problems according to feedbacks of my supervisor.

CFX and Mathematica results were approving each other. They were not same but they were close. That was the sign that the problem constructed correctly.

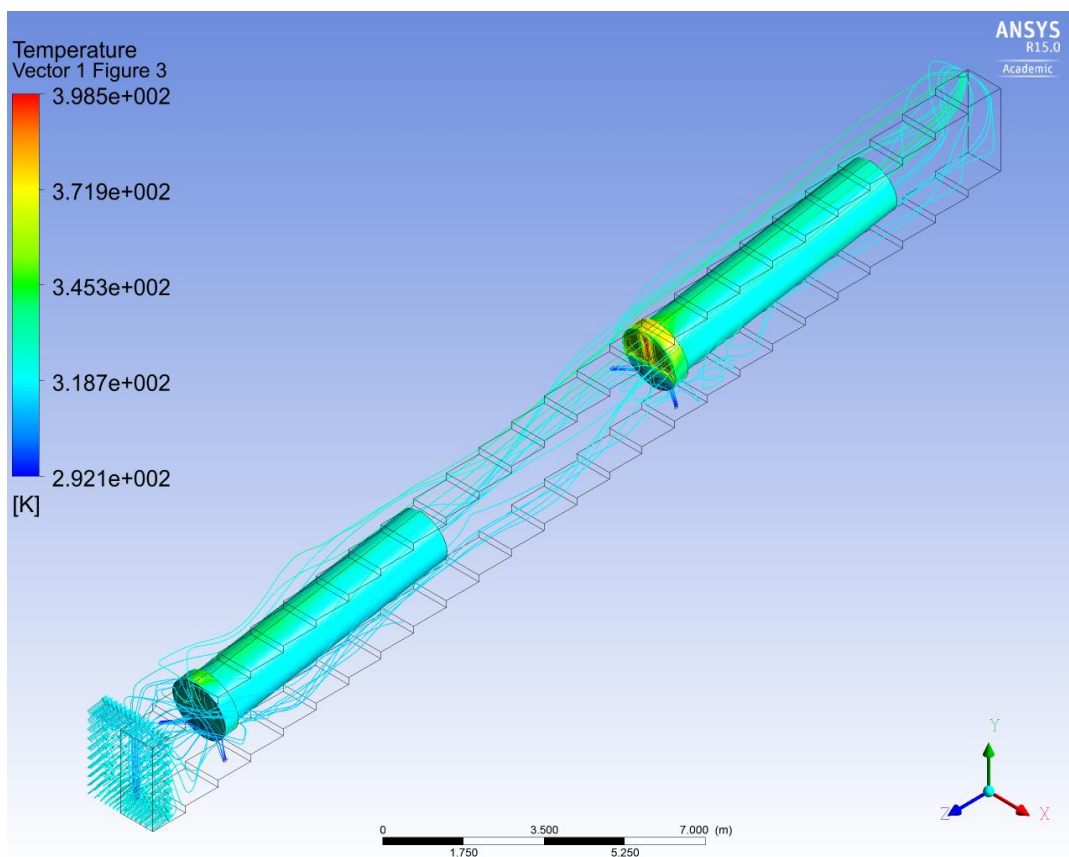
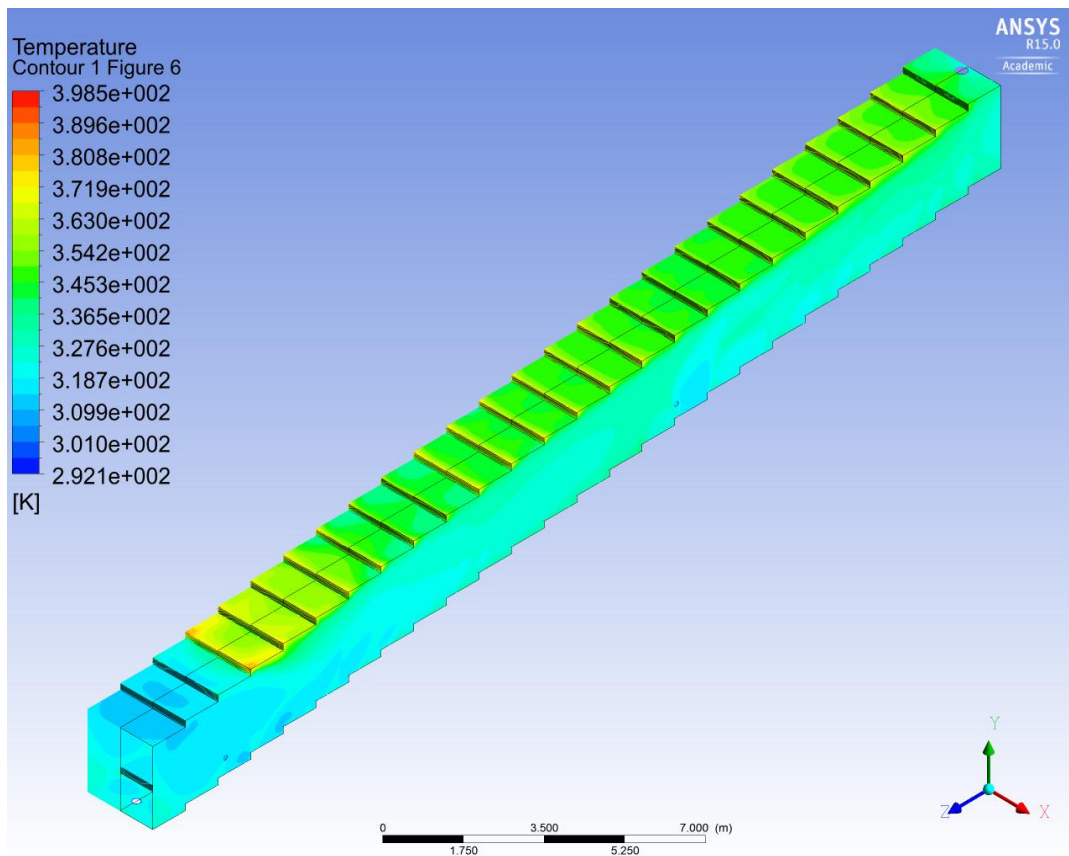


While the outlet temperature in CFX result was 330 [K], in Mathematica result (Temp.[Celsius] Vs Position[meters]) was 336[K].

I also see that the model that I tried to create more turbulence has indeed increased the film coefficient but since the huge sizes of the tunnel and low flow speed, it was almost insignificant.

As shown below, temperature is between logical values and desired film coefficient achieved. Maximum Temperature is 383K and total mass flow is 0.42 kg/s.

Inlet Conditions				
	Mass Rate	Temperature	Diameter	Inlet Speed
Flow that comes from decay pipe	2.100e-01 [kg s ⁻¹]	3.188e+02 [K]	2.2 m X 2 m	1.052e+00 [m s ⁻¹]
Chamber Inlet From Bottom	7.000e-02 [kg s ⁻¹]	2.928e+02 [K]	2.000e-01 [m]	1.259e+01 [m s ⁻¹]
Chamber Inlets From Sides X4	3.500e-02 [kg s ⁻¹]	2.930e+02 [K]	1.000e-01 [m]	2.501e+01 [m s ⁻¹]
Boundary Inputs	Heat Flux			
Chamber Walls	1.690e+02 [W m ⁻²]	-->40kW in total		
Target Front Part Heat Flux	1.208e+03 [W m ⁻²]	-->5.4kW in total		
Reflector Front Part Heat Flux	7.244e+02 [W m ⁻²]	-->3.2kW in total		
Results				
Outlet Temperature	3.284e+02 [K]			
max Chamber Surface Temperature	3.832e+02 [K]			
max Target Temperature	3.985e+02 [K]			
max Reflector Temperature	3.569e+02 [K]			
ave Heat Flux @ Chamber Surface	1.690e+02 [W m ⁻²]			
ave Heat Flux Coefficient@ Chamber	1.299e+01 [W m ⁻² K ⁻¹]			
max Heat Flux Coefficient@ Chamber	4.423e+02 [W m ⁻² K ⁻¹]			
max Heat Flux Coefficient@ reflector	7.705e+01 [W m ⁻² K ⁻¹]			
ave Heat Flux Coefficient@ reflector	2.777e+01 [W m ⁻² K ⁻¹]			
max Heat Flux Coefficient@ target	8.507e+01 [W m ⁻² K ⁻¹]			
ave Heat Flux Coefficient@ target	2.587e+01 [W m ⁻² K ⁻¹]			



Conclusion

The results show that helium's assist on cooling is undeniable. However, using only helium to remove the heat from the system would require a lot mass flow rate and it would cost much. Of course more calculations are required but as a first approach, the results show us it is doable. For helium part, working on streamlines, arranging them somehow not to leave a single hot spot are best things to do.

Some of the projects condition is represented below and can be found in
dfs\Users\i\igorgulu\Public

. *-----

Project 1

``Chamber and decay worst case 1123it 12.3hours``

--where the reflector and target 30kw and 50kw.

250m decay 40kw, 25m decay 10kw, chamber 40kw heat flow. 0.21kg/s decay inlet and
3X0.07kg/s chamber inlets.

``decay worst 16hours 6300it``--> only decay pipe.

-- 250kw and 70kw conditions for decay pipe. 0.105kg/s inlet for symteric solution. In full scale
0.21kg/s.

``decay chamber front heat 64hours ``

-- Reflector and target have heat source from front part as 3kw and 5kw (rest parts has 1kw)

250m decay 40kw, 25m decay 10kw, chamber 40kw heat flow. 0.21kg/s decay inlet and
3X0.07kg/s chamber inlets.

*-----

Project 2

``nodecayoutlet front heat 64 hours``

The model that does not have an outlet in decay pipe

-- Reflector and target have heat source from front part as 3kw and 5kw (rest parts has 1kw)

250m decay 40kw, 25m decay 10kw, chamber 40kw heat flow. 0.21kg/s decay inlet and
3X0.07kg/s chamber inlets.

*-----

Project3

``chamber decay best case 21.3hours 2200it``

--where the reflector and target 3kw and 5kw.

250m decay 40kw, 25m decay 10kw, chamber 40kw heat flow. 0.21kg/s decay inlet and 3X0.07kg/s chamber inlets.

``no outletdecay 44hours 3820it``

The model that does not have an outlet in decay pipe

-- Reflector and target have heat source 3kw and 5kw

250m decay 40kw, 25m decay 10kw, chamber 40kw heat flow. 0.21kg/s decay inlet and 3X0.07kg/s chamber inlets.

``decaybest 5600it 15 hours``

Symmetric solution.

250m decay 40kw, 25m decay 10kw (and 0.21kg/s mass flow in full scale)

**-----

Only Chamber

``Where the chamber has more details. (Symmetric)

-- Reflector and target have heat source from front part as 3kw and 5kw (rest parts has 1kw)

Chamber 40kw heat flow.

0.21kg/s inlet from decay part, 4X0.035kg/s chamber inlets from sides and 0.07 from bottom.