# Project :

# Fitting of the Goldak heat source model parameters based on FE Abaqus simulations

Made by:

El Houssaini Youness

Supervised by:

Florian Ditmann, Dipl. Ing.,Fraunhofer IWM

Master 2 CSMI

# Abstract

Within this report, one get to know the working enviroement in rhe fatigue and Fracture group where I conducted my project. Also the context of the problem in order to understand the outcome of this work which is a script automizing the whole parameter identification process. Controlling the goldak heat source model parameters based on finite element simulations is the main objective. The proposed script for that, is in cluding three main packages. The first one is in the postprocessing step of Abaqus simulation process. It is replacing the manual data extraction and its visualisation by an automatized workaround. Then, the package of processing the Abaqus FEA on the cluster which launches the job automatically and synchrenoze the results with the preprocessing stage by changing automatically the parameters. Most important, is the optimizer package which monitor the local machine processes and the HPC cluster results in order to iterate autonomously towards the optimal parameter identification. This is done using the gradient descent method minimizing a target function which takes into account the parameter magnitute and the main objective of having a matching FEA based temperature profile to the experiment measured temperature profile. Some results are presented and discussed as well. This automotized workaround is in its early stage and the results are as expected but still at least can be used as decision tool.

# Foreword

This report is part of the study program at the University of Strasbourg in the UFR Mathematics and informatics within the specialization of scientific computing and mathematics of information, Master 2 CSMI. This work done in the Semester Project subject supervised and examined by Pr. Christophe prud'homme. Thanks to whom this work had a foundamental start with her priceless advices. This work is considered as a preparation to the master thesis and also an opportunity to bring into practice all the theoretical lectures so far. It takes place at the Fraunhofer Institute for Mechanics of Materials IWM from the 1st October 2020 till the end of the semester, the 25th January 2021. The topic was accepted as a semester project thanks to the agreement of the study program (Master 2 CSMI) manager Pr. Christophe Prudhomme , to whom all my gratitude. He is also the examiner of this work. First and foremost I would like to thank the Fraunhofer Institute for Mechanics of Materials IWM that gave me this opportunity of evolving in both academic and professional scale. In addition, I would like to express my special thanks to my supervising professors, Dr. Vigon Vincent and Dr. Yannick Privat for the exemplary support by the University of Strasbourg. Also, my full gratitude goes to my project manager and supervisor Dr.Igor Varfolomeev and Dipl.-Ing.Dittmann Florian for their enduring support and helpful insight. Thanks to whom the working conditions and the atmosphere were extremely positive. Furthermore, my thanks to all my colleagues in the CSMI Master for the helpfulness and friendliness shown to me during my studies.

# Table of contents

# Chapter 1

# Introduction

## 1.1  Description of the envirenment

The Fraunhofer Institute for Mechanics of Materials IWM (German: Fraunhofer-Institut fr Werkstoffmechanik) in Freiburg, Germany, is a division of the Fraunhofer-Gesellschaft which focuses upon application-oriented research. The Fraunhofer IWM is a research and development partner for both industry and public institutions whose themes include the safety, reliability, life expectancy and functionality of materials in components and manufacturing processes. The broad spectrum of services offered by the Fraunhofer IWM's business units aid their clients and project partners in the assessment and continuing development of highly stressed materials and components and in the optimization of production process [?]. Research and development activities at the Fraunhofer IWM are organized into five business units where similar project topics are grouped together in each unit. This work is conducted in the "Component Safety and Lightweight Construction" group. Which is centered on the assessment of a component's safety and its fitness for purpose in terms of safety-relevant demands under operational loads. The applications range from proving the safety of power station components to confirming the fault tolerances of aerospace components, the life expectancy analysis of components in power stations and vehicles subjected to thermomechanical loads to crash analyses of vehicle components. The focus of this work is on welded components and their failure assessement based on the finite element analysis of the welding process heat input as explained below.

## 1.2  Context of the project

The failure assessment of welded components requires an accurate determination of the welding residual stresses in vicinity of the weld. Generally, the experimental determination of the WRS is not possible or very expensive and the analytical assessment routines of guidelines and technical standards are very conservative. This leads to a need for an accurate finite element analysis (FEA) of the welding process. Therefore, the FEA is split up into two parts, a heat transfer analysis covering the heat input of the welding process and a mechanical analysis calculating the nodal translations and stresses resulting of the thermal expansion and phase transformations. In the first part, the heat transfer analysis, the double ellipsoidal heat source model by Goldak [?] is used to simulate the heat input of the weld torch. The definition of the parameters of this artificial heat source is the main key for a realistic welding

simulation.

## 1.3 Objectves and sub objectives

The current way at the team where I am conductiong this project is a workaround where the parameters of the heat source are adapted and applied to simulations iteratively until the temperature-time history at defined positions outside the weld area agrees with experimentally measured data. This process is mainly based on expertise and can takes days if not weeks of fitting. In fact, the simulations can take hours for a single constalation of parameters and the decision for next iteration is not always obvious. The idea of this project is to automatize this workaround by a Python code that is iteratively adapting the heat source parameters for each weld pass. The code should automatically run in a loop to start Abaqus simulations, extract results, compare them to the target data and restart with adapted parameters of the heat source.
In order to reach the main objective, first of all one should do some Literature research:

- Study on welding process and welding parameters (e.g. linear heat inputs) to get into the topic.

Then Processing of the input data, which means:

- Extracting T-t curves from existing result files of FE simulations with properly fitted heat source parameters (this is taken as the experimental measurements, the target temperature to reach).

- Also, the existing data is of circumferential welds of austenitic steel pipes, but the method is independent of the material or geometry.

- And, arranging the input data to make it readable for the code.

Once this step is done then comes the step of Optimisation to identify the heat source parameters.

- Set up a Python code that automatically runs Abaqus jobs, extracts results, compares them to the target and restarts the FEA with adapted parameters.

And finally the Validation with experimental data of former projects [?].

## 1.4 Project and organization with realistic roadmap

The heat source model is based on litterature and has 4 main parameters. Those parameters are Q: the heat source maximum intendity, the dimension of the paraboloid temperature distribution. At first try and to simplify the problem in order to understand better the governing models, only Q is considered as parameter. The other parameters are fixed to certain value. In order to find the optimal parameter Q, one should define an objective function based on extracted data as explained above. The purpose is running automatically a number of simulations given predefined parameters. The parameters are found in the input file and are modified based on a python script. The material model file is also required to

run the main script launching the job on the computation server. Once the simulation is done, a post-processing script has the task to extract the predefind measures of temperature according to the target function variable.

In order to reach those objectives in time a Gantt diagram is synchronized with github issues, milestones and deadlines. This is a glance of how my work is organized so far and how Gantt viewer is representing my work packages.



Figure 1.1: A screenshot of my Github project Gantt viewer.

The work on those packages was done in a regular basis, and here a zoom on the left part of the above gantt diagram. There was a disconection at the end of the semester due to other projects and exams which cost a report delay. Also, I did not anticipate all the packages because there was a considerable part devoted to analysing results which i did not have enough time to finish properly. Still creating a roadmap and puting it on Gantt diagram helped a lot to have controle over my work.

| Name | Assignee |
|---|---|
| ⊟ ue-projet-issues | (None) |
| #9 T-t profile extraction script | youness-elh |
| #10 Running abaqus jobs script | youness-elh |
| #11 Objective function | youness-elh |
| #12 Algorithm | youness-elh |
| #13 install the action runner in the server | youness-elh |
| #14 Run first action on self hosted runner | youness-elh |
| #15 Ask for more models | youness-elh |
| #16 Sphinx/Antora | youness-elh |
| #17 Add gantt in the report | youness-elh |
| #18 Modeling the optimization problem | youness-elh |
| #19 Convergence and assessment of the algo | youness-elh |
| #20 Enhancements | youness-elh |
| #8 Workpackages + Gantt generation | youness-elh |
| #4 Make a summary | youness-elh |
| #6 Fill the project objective section | youness-elh |
| #5 Fill env & context subsection | youness-elh |
| #7 Problem compiling .tex file (related to issue #2) | prudhomm |
| #2 Continuous TeX Document Production with GitHub Actions | youness-elh |
| #1 Start the first version template | youness-elh |
| #3 make a roadmap | youness-elh |

Figure 1.2: A Zoom on the issues of my Github project Gantt viewer.

# Chapter 2

# State of the art

In this chapter, we will have an overview of the current methodology used to identify the Goldak heat source model parameters.

## 2.1  Current procedure

Welding process finite element based models are caliberated based on the derived measurements as shown below.



Figure 2.1: Temperature measuring during welding process on several points [5].

The obtained temperature profile based on the aboce experements are considered as a

reference to compare the FE models with in order to identify the key parameters. This might be useful for caliration, validation or also calculating the parameter deviation. Here is an example of temperature profiles of a FEA based model compared to the experiments temperature measurements.
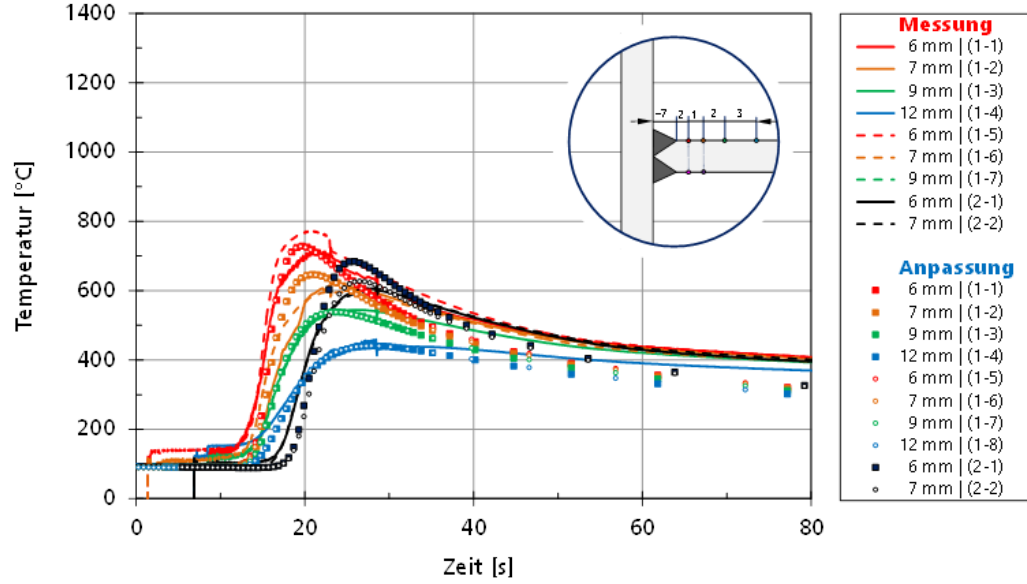


Figure 2.2: Temperature history for both experements and FEA based model [5].

In order to obtsin such a graph, the adopted method so far is based on the Abaqus graphical interface and extracting the profile manually. This is labor intensive especially when one has to wait around 3 hours and check regularly before starting the simulation temperature profiles extraction.

## 2.2 Detailed used model (Goldak)

Welding is a fabrication process whereby two or more parts are fused together by means of heat, pressure or both forming a join as the parts cool. Welding is usually used on metals and thermoplastics but can also be used on wood. Moving heat sources is a topic in transient heat transfer that is applicable to engineering problems, particularly welding engineering. In the early 20th Century, welding engineers began studying moving heat sources, both empirically and theoretically [1]. The Goldak heat source model is adopted to simulated welding process as a moving heat source mapped as follows:

$$Q(x,y,z) = \begin{cases} Q_0 exp\left(-\left(\dfrac{x^2}{b^2} + \dfrac{y^2}{c^2} + \dfrac{z^2}{a_f^2}\right)\right); x \geq 0 \\[4mm] Q_0 exp\left(-\left(\dfrac{x^2}{b^2} + \dfrac{y^2}{c^2} + \dfrac{z^2}{a_r^2}\right)\right); x < 0 \end{cases}$$

Figure 2.3: Goldak heat source distribution [4].

it is implemented to the Abaqus simulation via USER subroutine DFLUX Double-ellipsoidal heat source. $(,,)$ is the Gaussian distribution of the power density where $Q_0$ is the maximum value of the power density at the center as presented in the figure below.
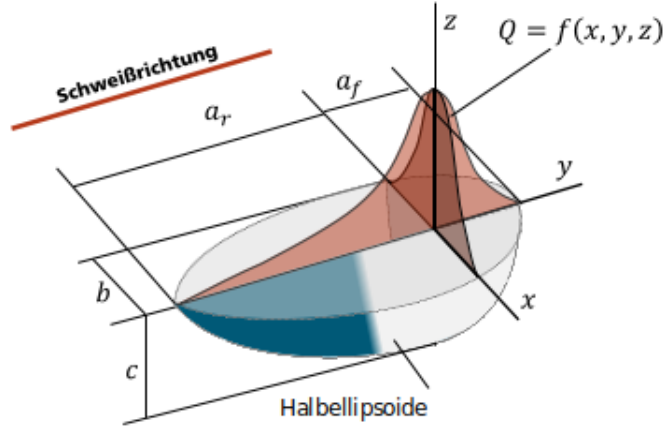


Figure 2.4: Representation of the Goldak Double-ellipsoidal heat source [5].

The parameters to identify are thus:

$$a, \; b, \; a_r, \; a_f \; and \; Q_0$$

In this project, only the maximum value of the power density is considered. Beside, b and c are given parameters as they define the weld bead geometry. Also, $a_r \; and \, a_f$ are dependant. The next step is taking one of them into consideration in our optimization problem.

# Chapter 3

# Optimizer tool

The following chapter is devoted to explane the outcome of this project which an optimization tool working iteratively, autonomosly and finding the optimal Goldak parameter to match a target temperature profile.

## 3.1   Principle

The principle of such tool is replacing the labor intensive work of watching over the Abaqus process and iterating acordingly with new trial of parameter. This process can be represented as follow:
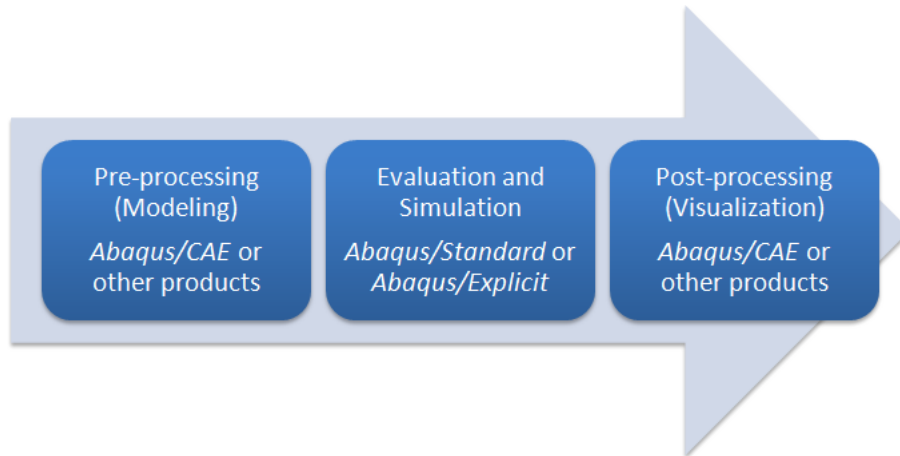


Figure 3.1: Abaqus FEA software products used in Finite Element Analysis [1].

For such purpose we establish the following cost function which is Q dependent. The minimum is met for the optimal value of Q.

With:

The chosen algorithm to minimize the mentioned cost function is based on the gradient

$$Minimize_v \ Loss(Q)$$

$$\text{With} \ \ Loss(Q) = \frac{1}{2}|Tsim - Tref|_{L^2}^2 + \frac{\alpha}{2}|Q|_{L^2}^2$$

- ✓ $\alpha$ Is a regulation and penalization parameter.
- ✓ Tref is the reference temperature profile.
- ✓ Tsim is the simulation profile dependent on Q.

descent method.

## 3.2  Implementation
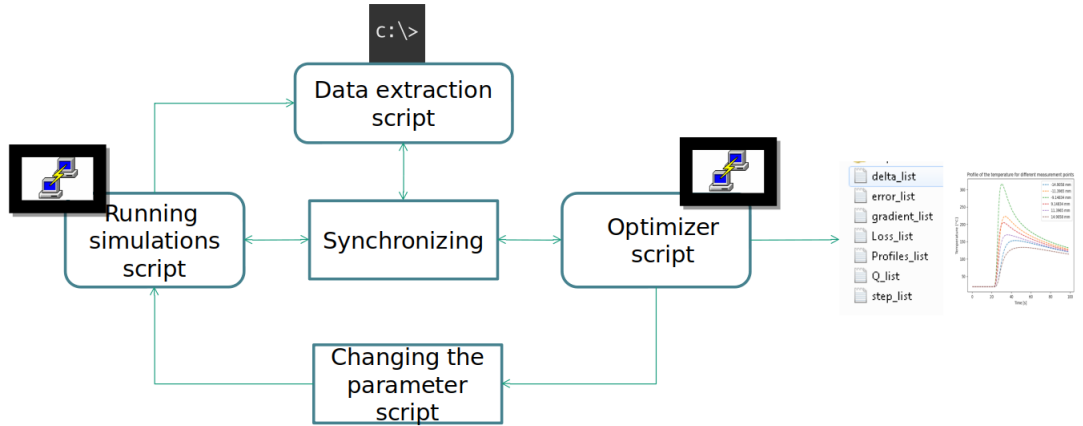
The optimizer tool can be sketched as follow:



Figure 3.2: The automotized workaround to find optimal parameter.

Beside the obvious packages for such process explained above, synchronizing the packages is a key package in this proposed workaround. Also, the optimization algorithm is of a big importance, especially the nested loop to look for a descent direction to pull back the values of he parameter when diverging.

```
while ((count < max_iter_step)  and (delta >=0)):
    print('------------------------------------------------------------')
    print('-----Looking for descent direction for iteration '+str(count+1)+'------')
    print('------------------------------------------------------------\n')
    step /= 2.3
    Q = max(Q_old - step*gradient,0)
    profiles = Abaqus(Q)
    L_new = Loss(Q,profiles,Targets)
    delta = L_new - Loss_list[-1]
    count +=1
    #save
    Q_list.append(Q)
    Loss_list.append(L_new)
    delta_list.append(delta)
    step_list.append(step)
    Profiles_list.append(profiles)
```

Figure 3.3: The descent direction loop.

The tool is implemented and is actually working in both the cluster for Abaqus simulation and local machines for pre and post processing.
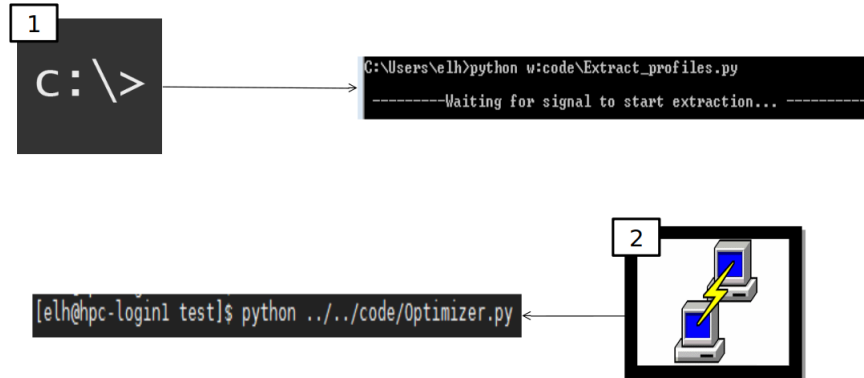


Figure 3.4: The way the optimizer tool is executed.

## 3.3   Tests and results

Every Abaqus simulation is running around 3 hours and the tool might exceed 8 iterations before converging which make it already 1 day. So, here the few results and tests I made so far. please note that the tool is not yet validated so the following cases are not for validation but just a proof of operationality. in the meantime, some comment might be given.

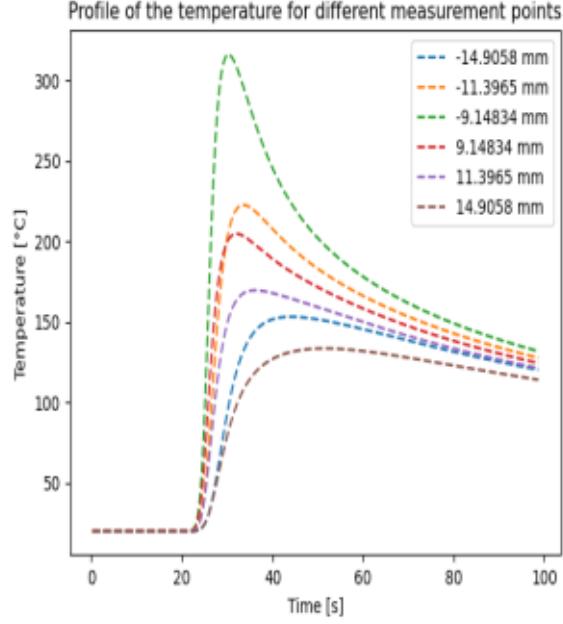First we define the target profile as follows:



Figure 3.5: The target temperature profiles.

In order to optimize the calculation time especially in this exploration stage of the tool we consider only 40% of the whole welding process duration.
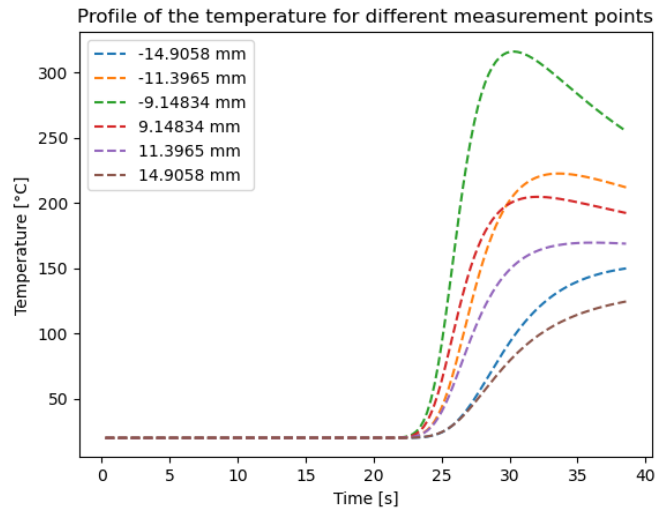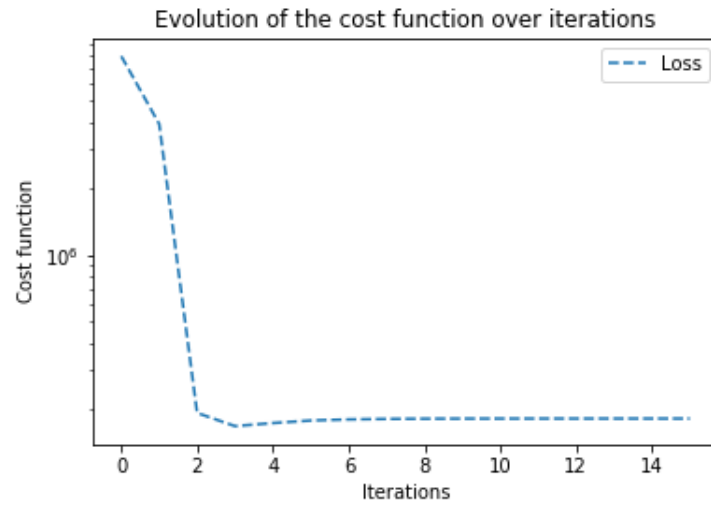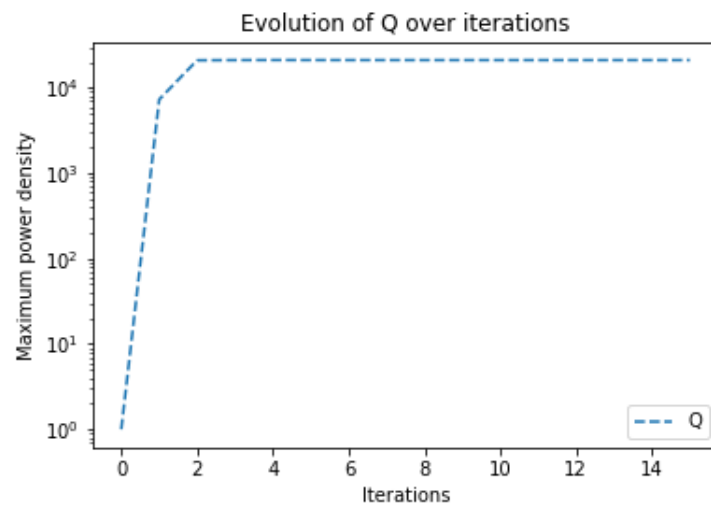


Figure 3.6: The less time consuming target temperature profiles.

With this hypothesis and a penalisation/regulation factor $\alpha = 10^{-14}$ we have a coverging sequence as shown below with the cost function evolution over iterations.

Evolution of the cost function over iterations

And accordingly the optimized parameter is as follows.



Evolution of Q over iterations

The result seems promissing but the temperature profiles are not as expected. See the figure below.
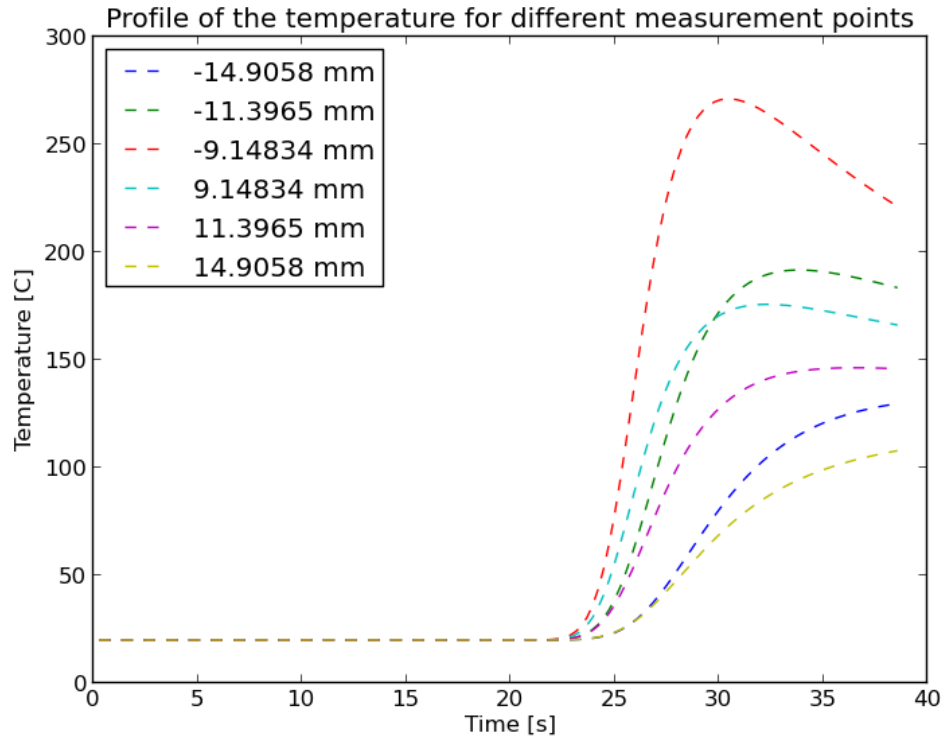


However the target profile is higher thatn 300 C as one can see below.
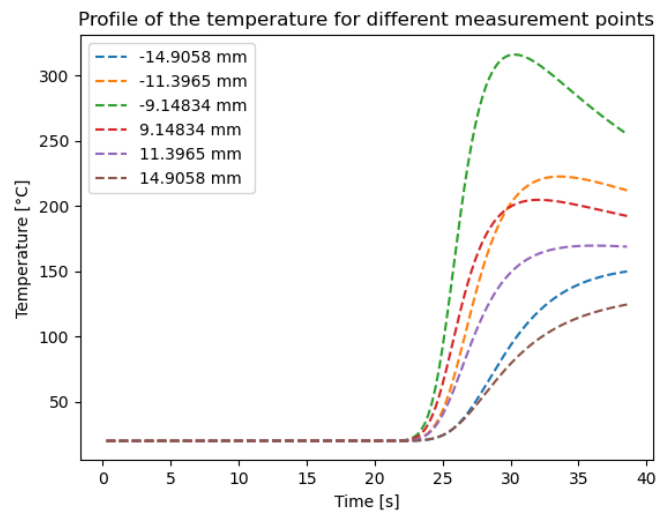


Figure 3.7: The less time consuming target temperature profiles.

# Chapter 4

# Conclusion

The optimizer tool has reduced in a big part the labor intensive work of the Abaqus modeling, simulating and postprocessing. Regarding, the Goldak heat source parameter, the results are promessing but still not satisfying. One idea is to look for better penalisation/regulation coefficient ($\alpha$) and also thinking about normalizing the loss which I did not do so far. The tool is really at its early stage and still a lot to analyse namely the tolerance, magnitute of values i.e Q and T. Once the tool is running perfectly we can think about including a second parameter or more. To have a look to the code one can check my repository branch: optimizer tool.

# Bibliography

[1] Abaqus software.

[2] Fraunhofer institute for mechanics of materials.

[3] Welding.

[4] J. Goldak, A. Chakravarti, and M. Bibby. A new finite element model for welding heat sources. Metal. Trans., 15B:299–305, 1984.

[5] M. W. I. V. W. Rudorffer, F. Dittmann. Modellierung von schweinhten zum nachweis der ermdungsfestigkeit mit dem rtlichen konzept. Abschlussbericht zum IGF-Vorhaben, Nr. 20.025 N,:299–305, 2020.