BRUNEL UNIVERSITY

Electronic System Modelling of UT Pulser-Receiver and the Electron Beam Welding Power Source

By
Thayaparan Parthipan

A thesis submitted in partial fulfilment for the degree of Engineering Doctorate

In the
School of Engineering and Design

September 2013

ABSTRACT

Continuous improvements to industrial equipment used in essential industrial applications are a key for the commercial success to the equipment manufacturers. Industrial applications always demand optimum performance and reliability and almost all equipment used in industrial applications is complex and are very expensive to replace. Often modifications to hardware and retrofitting additional hardware are encouraged by most equipment manufacturers and operators. The complexity of these systems however, makes assessment of modifications and design change difficult. This research implemented system modelling techniques to overcome this issue, by developing virtual test platforms of two distinctive industrial systems for enhancement assessment. The two distinctive systems were the electronic equipment called pulser-receiver used in ultrasonic non-destructive testing of safety critical oil & gas pipelines and a high voltage power supply used in high energy electron beam welding. Optimisation with emphasis on portability of the pulser-receiver and rapid weld recovery after a flashover fault condition in the electron beam welding application required assessment before design changes were made to hardware. SPICE based simulators LTSpice and PSpice were used to model and simulate the pulser-receiver and the welding power supply respectively. All the models were evaluated appropriately against theoretical data and published datasheets. However, validation of low level component models developed in the research against measurement data at a component level suffered due to system complexity and resource constraints. Close mapping of simulation results to measurement data at a system level were obtained. The research helped build up a wealth of knowledge in the development of circuit simulation models that can be analysed in the time domain with no non-convergent issues. Simulation settings were relaxed without compromising accuracy of model performance.

ACKNOWLEDGEMENTS

The author extends his eternal gratitude to all those who have helped in the project, at Brunel University, the University of Surrey and TWI ltd. The work could not have been completed without the help and support of the project supervisors Dr. Rajagopal Nilavalan, Professor Wamadeva Balachandran, Colin Ribton and Professor Peter Mudge, as well as that from Dr. Mohamed Darwish, Barry Elborn and many other colleagues too numerous to mention here. The continued camaraderie from other Research Engineers on the Brunel/Surrey EngD Programme, both past and present, has also been an essential ingredient and is very gratefully acknowledged, as is the permanent support and encouragement from family and friends.

Special thanks go to Dinesh Chamund at Dynex Semiconductor, Peter Johns from International Transformers and Mark Mayes from Linear Technology (Arrow Electronics) for all the materials they provided towards the research. For the project funding, thanks and recognition must also be apportioned to The Engineering and Physical Sciences Research Board (EPSRC) and TWI Ltd., without whom the research would not have been possible.

I also would like to thank my wife, Kavita for her constant support and encouragement, my son Kathiravan for proof reading some of my work and my daughter Kartikaa for helping me to keep things in prospective.

To my mother, father, uncles and aunts

CONTENTS

LIST OF FIGURES

LIST OF TABLES

List of tables

ABBREVIATIONS

ABM
Analogue Behavioural Models
AC
Alternating Current
ADC
Analogue to Digital Converter
CMOS
Complementary metal-oxide-semiconductor
DAC
Digital to Analogue Converter
DC
Direct Current
DDR2
Double Date Rate synchronous memory
DSN
Distributed Sensor Network
EBA
Electron Beam Acceleration
EBW
Electron Beam Welding
EDA
Electronic Design Automation
EMI
Electro Magnetic Interfurance
ENOB
Equivalent number of bits of an ADC
FEA
Finite Element Analysis
FPGA
Field Programmable Gate Array
FWD
Free Wheel Diodes
HT-Bank
Bulk capacitor bank used in the pulser receiver
HTPSU
High Volrage power supply used in pulser-receiver
IC
Integrated Circuits
IGBT
Insulated Gate Bipolar Transistor
I-V
Current-Voltage characteristics

LASER
Light Amplified by Stimulated Emission of Radiation
LRUT
Long Range Ultrasonic Testing
NDT
Non-Destructive Testing
nMOS
n-channel MOSFET transistor
NPT IGBT
Non-Punch-Through IGBT
NVEBW
Non Vacuum Electron Beam Welding
PC
Personal Computer
PCB
Printed Circuit Board
PHY
Physical layer of the Ethernet controller device
PM
Phase margin
PRR
Pulse Repitition Rate
PSRR
Power Supply Rejection Ratio
PT IGBT
Punch-Through IGBT
PZT
Lead Zirconate Titanate piezoelectric transducer
SMPS
Switch Mode Power Supply
SPICE
Simulation Program with Integrated Circuit Emphasis
TWI
The TWI Ltd.
UT
Ultrasonic Non Destructive Testing
VHDL
Very high speed Hardware Description Language

LIST OF SYMBOLS

A	Area - m^2
C_{55}	Elastic stiffness constant - N/m^2
C_{LOAD}	Capacitive load
C^T	Free capacitance of an piezoelectric transducer measured at 1 kHz - F
d_{15}	Piezoelectric charge constant - C/N
dB	Decibel
e0	Permittivity of free space - C/Vm
$E_{11}{}^S$	Piezoelectric stress constant under constant strain - C/N
$E_{11}{}^T$	Piezoelectric stress constant under constant stress - C/m^2
f_a / fp	Antiresonance or Parallel resonance frequency - Hz
F_{ext}	Frequency of excitation signal used for transducer excitation - Hz
f_r	Resonance frequency - Hz
g_{15}	Piezoelectric voltage stress constant - Vm/N
G_{PA}	Gain of power amplifier
I_d	Drain voltage of MOSFET - A
k_{15}	Shear coupling factor
Qm	Mechanical quality factor
$S_{55}{}^D$	Elastic compliance coefficient under constant electric displacement - m^2/N
$S_{55}{}^E$	Electric compliance coefficient under constant electric field - m^2/N
v	Velocity of an electron - m/s
$V_5{}^D$	Sound velocity - m/s, 1

V_{CE}	Collector-Emitter voltage across
V_{KVOUT}	Acceleration voltage
V_p	Sound velocity (also labeled as V_5^D) - m/s
V_r	Relativistically-corrected acceleration voltage - V
W_b	Power density - kW/cm^2
X_C	Impedance of a capacitive load
π	3.141592654, 1
ρ	Density - kg/m^3

1 INTRODUCTION

1.1 Introduction

Better understanding of the operation and performance of systems and their constituent components is crucial when it comes to adapting hardware changes or modifications. Such changes are required mainly for performance enhancement of the overall system or at a component level. Components may be a single device or may be a subsystem consisting of a number of individual devices.

Performance of the constituent components and the interaction between them varies for different functionalities and scenarios in most systems and this variation consequently affects the overall performance of the system. These components' functionality in general is electronic, mechanical, chemical, or acoustic or a combination of more than one of these aspects; making the component and hence the system a multi-engineering discipline. This increases the complexity of system analysis and makes it an arduous and time consuming task to analyse and optimise the systems in their entirety.

Modelling the systems and simulating the developed model in a computing environment has been proven as a flexible means of achieving this aim efficiently. Validation and assessment of systems and components by system modelling has been extensively published in the literature. Some of the work sited in the literature relates to system modelling as follows: In avionics and the space sector, being conservative in their design approach (as many failures have a safety implication), the assessment of components is a must. A systematic approach of modelling and simulating avionics hardware and software for assessing components at a subsystem level was

discussed by (Bluff 1998), and examples such as analysis of aerospace circuit fault analysis by (Signal & Somanchi 1998), performance analysis of improved power electronics for aircraft ventilation and air-conditioning motors (Athalye 1999) at a component level and battery life assessment of space probe by (Michael 1998) are evidence of the powerfulness of the system modelling techniques. Likewise in automotive engineering, where reliability and cost are paramount, system modelling has been used extensively to qualify automotive subsystems such as works published by (Ryan & Philip 2008); estimating battery capacity for an electric vehicle, (Donnelly & Gauen 1993); a high power switching device evaluation on an ignition system, and (Metzner, Schafer & Chihao 2001); looking at modelling mixed-signal electronic components used in an automotive application for performance analysis. Evidence of utilising system modelling techniques for foreseeing uncertainties in a distributed sensor network (DSN), that mostly fails prematurely due to a lack of power, has also been sighted in the literature (Wei 2007), (AboElFotoh, Iyengar & Chakrabarty 2005).

The Long Range Ultrasonic Testing (LRUT) pulser-receiver unit and the Electron Beam Welding (EBW) power source are two distinctive industrial systems. The LRUT pulser-receiver unit is used in ultrasonic Non-Destructive Testing (UT) applications for structural health monitoring of operational sensitive components. The EBW power source is a high power Switch Mode Power Supply (SMPS), which provides high power (100 kW) for precision EBW of high value components. Reliable and optimal performances of these complex systems are critical for the smooth operation of industrial applications, LRUT and EBW. The research focussed on developing simulation models of these two distinctive industrial systems, which can be simulated in a computer environment for performance analysis and subsequent modification to their hardware for their enhanced performance.

1.2 Long Range Ultrasonic Testing technique

Industry needs reliable cost effective technologies for continuous or scheduled structural monitoring of structures. UT is a method of inspecting the condition of structures and components using ultrasonic waves. This method ensures that the material and its mechanical properties are not damaged or compromised. LRUT is a subset of UT that has the ability to detect defects for many metres in different shapes and type of structures such as pipes (Gharaibeh 2008), (Mudge, Lank & Alleyne 1994), rails (Gharaibeh et al. 2010) and plates (Mudge & Catton 2006). This technique enables the inspection to be carried out from a single point of access if required (Parthipan 2010).

The effectiveness of this technique has broadened its applications, from pipe inspection to inspection of various complex and vast structures such as offshore wind-turbine towers and offshore platforms. These structures mostly operate in remote, harsh environments. Compliance with international standards, aimed at ensuring reliability and safety (e.g., ISO 19900 and ISO 19902) whilst operating in a harsh environment requires periodic condition monitoring of structures. An electronic portable pulser-receiver unit facilitates application of the LRUT technique on site. The unit uses ultrasonic waves to detect anomalies by sending ultrasonic waves along the specimen and then processes the ultrasonic waves received. The received signal can either be a reflection from the defects when operated in the pulse-echo method or the shadowed image after the defects in the pitch-catch method.

1.3 Electron Beam Welding application

EBW is a welding process in which a high velocity electron beam is applied to the materials being joined. The process melts the work piece by transforming the kinetic energy of the

electron beam into heat upon impact. It is very popular in industrial applications such as aerospace, marine, nuclear waste burial and automotive for its high integrity, low distortion, higher welding speed, lower heat input, and greater depth-to-width aspect ratios than any of the other fusion welding methods (Fritz et al. 1998), (Russell 1981), (Schulze & Powers 1998). A high voltage potential (150 kV in typical applications) is required between the welding electrodes (Electron beam gun anode and cathode) to gain high velocity. The required voltage level is achieved using a transformer-rectifier fed with a single phase from a switch mode inverter unit, based on an H-Bridge inverter.

1.4 Motivation

Electronic systems that are deployed on industrial applications in general incorporate complex hardware. Their reliability and optimal performance is a key factor for their commercial success. Typically these systems are a fabrication of a number of subsystems, whose function differs from each other, but which are inter-dependent on each other. This makes fault diagnostics for maintenance and implementing modifications for enhancement complicated. Moreover, the application specific nature of their design and operation always requires expert knowledge when it comes to repair, maintenance, enhancement and usage of this hardware. This adds cost and intensive resource usage for the equipment developers, with a negative impact on maintenance downtime and product development cycle time.

Electronic system hardware that facilitates the LRUT technique and EBW applications are complex architectures. Consequently when examining the hardware for maintenance or enhancement, or when making changes to the system for trials, there is a risk of the system being damaged and/or equipment safety being compromised. However, continuous improvements to

find and implement modifications for reliability, and update performance enhancement is necessary for equipment manufacturers and service providers to maintain a lead. Therefore, a reliable method to appropriately analyse these complex systems is required, so that maintenance and product development can be carried out quickly, economically and safely.

1.4.1 Pulser-receiver optimisation for portability

A battery powered portable electronic equipment generally known as a pulser-receiver (see section 2.3 for further information) is utilised in the non-conventional UT of oil & gas pipelines. These pipelines operate in extreme operational conditions and are in general installed in remote and harsh environments. Portability of the inspection instrumentation is paramount for the ease of use and installation. The existing pulser-receiver equipment requires enhancement, with emphasis on portability without compromising its functionality for optimum usage.

1.4.2 Electron beam welding power supply enhancement for void-free welding

Electron beam welding uses a very high voltage (150 kV DC voltage in a typical application) in the welding process (Sanderson 1986), (Schulze & Powers 1998). This, when the vacuum integrity between the weld electrodes is jeopardised, causes a high voltage breakdown. This undesirable scenario is called flashover and it can do damage to the expensive high voltage power source and the valuable weld specimen. When flashover is detected, fault detection circuits implemented in the system temporarily terminate welding by halting the inverter operation for a set time referred to as a weld dead time. This action prevents damage to the expensive power source and high value components. This weld dead time however, if too long may cause voids in the weld which are unacceptable in high quality welding. This leads to a study on enhancing the welding process, with emphasis on flashover fault detection and fault

recovery control regime implementation. In particular it was explored whether reducing the duration of the weld dead time would be possible with the existing power source hardware.

1.5 Scope of the thesis

The aim of this thesis is to provide simulation test platforms that can be used for analysing electronics systems (pulser-receiver and the EBW power source) and testing modifications aimed at improving their performance. The pulser-receiver used in the LRUT application and the associated tools form a multi-engineering discipline system which requires high end modelling tools with expert knowledge on all aspects of its system if one wants to model it for analysis. In general, acquiring high end tools and providing expert knowledge is not always possible due to project financial and time constraints. Hence a simplified test platform was required where application specific issues can be simulated for the enhanced portability of this LRUT instrumentation.

On the other hand, the power source and its control electronics that provides welding power for electron beam welding, is a complex electronic hardware with a number of sub systems with inter-dependent functionalities. Enhancement of the system for optimum parameter settings for fault detection and fault recovery time after flashover detection requires analysis of functionality at a component level. This is almost impossible with this complex system due to access restrictions and inseparable functionality between components and sub systems. A reliable method of analysing the consequences of parameterising the system to give an enhanced operation was required so that the impact on individual components and subsystems due to parameterisation can be investigated.

System modelling and simulating the systems at both a system level and at a component level was chosen as a method to form a virtual test platform for both the LRUT system and the EBW power system. The following objectives were set to achieve this goal;

- Development of pulser-receiver simulation model for investigating possibilities of enhancing portability without compromising its power and functional performance.
- Prototyping of power electronics for the pulser-receiver for enhanced performance and portability.
- Adaptation of the developed pulser-receiver model for other variants of applications.
- Development of an EBW power source simulation model that can be simulated for functional analysis of the power stage and its control electronics with dynamic load conditions.
- Analysis of fault recovery control regime implemented in this EBW power source.
- Stability analysis of this EBW power source.
- Investigation of loading effects on this EBW power source.

1.6 Circuit simulation using SPICE based program

Modelling and simulation of systems are in general carried out at a device level for device characterisation and at a circuit level for assessing the circuit and the overall system performance. The components of the LRUT system are electronic equipment - pulser-receiver and electro-mechanical sensors - piezoelectric transducers. The enhancement of its hardware as will be discussed in Chapters 2, 3 and 4 is linked to modification and re-design of some of its electronic hardware. Its functionality and construction heavily depends on the behaviour of its load (piezoelectric transducers) and the way they are excited. The EBW power source hardware

comprises high voltage power electronics and low voltage control circuits. In essence, both systems are considered electronic hardware in spite of some of the constituent components having a non-electrical functionality. These non-electrical components in these systems, however, act upon the application of an electrical field on their electrical domain ports (e.g., electro-chemical: battery cells; electromechanical: piezoelectric transducers; and thermionic: EBW – Gun cathode).

SPICE (Simulation Program with Integrated Circuit Emphasis) is a modelling language/ programme originally aimed at fulfilling the needs of simulating and analysing mainly Integrated Circuits (ICs) (Christophe 2008), (Vladimirescu 1990). It evolved into a number of different versions such as SPICE1, 2 and 3 and became a main computer aided analysis program used in the design and analysis of analogue circuits (Vladimirescu 1990) and lately for simulating mixed signal circuits (Vladimirescu 1999). Review of its evolution is very well documented in (Vladimirescu 1990) and (Vladimirescu 1999).

SPICE's inherently efficient ability of accurately predicting waveforms of circuits has made it popular amongst the electronic circuit design community, whose applications always require components to be connected and simulated for assessing performance of the components as well as the connections between components. SPICE however does not permit circuit schematics to be generated.

1.6.1 Simulation tool selection

There are handful of Electronic Design Automation (EDA) tools that allow circuit schematics and models to be generated/ developed and simulated in a computer environment. Four such tools had been identified as the most suitable candidates for the system modelling and

simulation of LRUT pulser-receiver and the EBW power source: System-Vision (Mentor Graphics 2012), Saber (Synopsys 2012), PSpice (Cadence - PSpice 2012) and LTSpice (Linear Technology - LTSpiceIV 2012). These tools have SPICE adaptability and allow schematics to be generated for simulation.

The general review of using these tools together with pros and cons associated with them are documented in (Moreland 1998) and (Parthipan 2010). System-Vision and Saber are high-end industrial leading multi-engineering discipline simulation tools. These tools allow multi-engineering discipline systems to be modelled and simultaneously simulated in a single platform. Simulating all constituent components in the system simultaneously in a single platform allows the coupling and inter-dependant nature between components to be studied effectively. The licence fee for these tools is very high (in excess of £5000 for a commercial licence) and is not always justified by the project needs. These tools, because of their precise modelling capability for accurate modelling and simulation, require expert knowledge in all relevant engineering disciplines that the constituent components possess. Computing power requirement is also high, because of the complexity of the fully featured models.

PSpice is a high end SPICE-like modelling and simulation program, optimised for electronic circuits and components simulations with a moderate licence fee (approx.. £2000). It allows electronic schematics to be generated and it also supports mixed-signal simulations; both analogue and digital circuits can be simulated simultaneously in a single platform. PSpice is supplied with SPICE based component models of most commonly used discrete and semiconductor ICs from most major semiconductor devices manufacturers and allows third party SPICE based models to be imported and simulated.

LTSpice is a manufacture specific SPICE based simulation program, mainly aimed at simulating power products manufactured by Linear Technology (Linear Technology - LTSpiceIV 2012). This program is supplied with topology specific models of Linear Technology power products only, that can only be simulated in LTSpice. LTSpice, like PSpice also allows third party SPICE based models to be imported and simulated, but only supports analogue simulation. It is a licence free simulation tool.

This work utilised LTSpice for the modelling and simulation of a LRUT pulser-receiver for the reason explained in Chapters 2 and 3, the enhancement process required Linear-Technology power products to be utilised and simulated for rapid prototyping and foreseeing uncertainties. Moreover, the project had constraints associated with finance and expert knowledge. PSpice was not an option due to the unavailability of SPICE models for Linear-Technology power products used in the circuits.

The simulation model of the EBW power source was developed using PSpice, because models for almost all main constituent components in this system were available in SPICE or could be developed in PSpice. Moreover, there was a necessity for including digital and mixed-signal component models for the entirety of the system model, which was impossible with LTSpice. Multi-engineering domain EDA tools were not required for this work.

Complex and precise models are not always needed to produce sensible simulation results. Abstract models with relevant component information are equally useful in simulation for producing useful results. These equivalent circuit models are in general less complex and only include the component parameters specific to the functionality of interest and most importantly allow other engineering discipline components to be modelled with acceptable

accuracy. This research work assumed 80% accuracy to be adequate for modelling constituent components that poses functionality other than electrical/ electronic (see sections 3.2.2 and 3.5 for explanation). This work also assumed that the accuracy level of the component models outside the operational range of the real component is less significant.

1.6.2 Simulation models

Models published by the components manufacturers are mostly built-in with component specific parameters and parasitic components which make the model close to real components. These models are called topology specific models. PSpice model libraries provided by its current vendor (Cadence - PSpice 2012) provide a large collection of component models from all main semiconductor manufacturers, but do not provide topology specific models for all devices on the market. However, the tool allows third party SPICE based models to be imported and simulated.

PSpice also allows users to develop equivalent circuit models of components using its primitive Analogue Behavioural Models (ABM). ABM models help modelling of less complex simulation models (Cotorogea 1998), and other engineering discipline models such as acoustic wave behaviour (Aouzale, Chitnalah & Jakjoud 2009), acoustic transducers (Parthipan et al. 2011) and battery chemistry (Gold 1997).

The LTSpice model library supplied by its vendor, Linear Technology, only consists of models of power products manufactured by them and like PSpice, LTSpice also allows third party SPICE based models to be imported and simulated. It also supports equivalent circuit models developed using SPICE-like primitive components.

1.6.3 Transient analysis

LTSpice and PSpice support nonlinear transient analysis, nonlinear DC analysis and linear small signal AC analysis. Both linear and nonlinear circuits can be analysed. The non-linearity is very common in complex circuits, which is mainly caused by the nonlinear current-voltage characteristics of semiconductor devices.

Transient analysis is a time based analysis of systems over a finite interval. This permits voltage and current waveforms, which appear at the nodes of the circuits, to be analysed for non-stability issues, which are very common in non-linear circuits. Almost all power electronics and modern ICs are non-linear due to their dis-continuous (switching) functionality and the nonlinear current-voltage (I-V) characteristics of the semiconductor circuits. Circuit functionality related issues such as ripple levels, conduction losses, switching losses, effects of parasitic elements introduced by components, EMI, settling time and instability issues that commonly cause poor performance can be assessed using this mode of analysis. Appropriate transient models included with parasitic elements are required for this analysis. With the utilisation of correct models that represents the components and interconnections a virtual test platform that reflects reality can be created (Christophe 2008).

This research work mainly focussed on the effects commonly associated with nonlinear and time varying circuits such as distortion and power consumption which involve transient related analysis. The research, however, included some linear AC analysis (small signal analysis) for interpreting frequency response of components that are exposed to signal waveforms of a range of frequencies and of variable amplitude. Feedback control circuits and signal conditioning circuits which required steady state analysis were examples that underwent linear AC analysis.

1.6.4 Convergence issues

Nonlinear circuits and components are prone to convergence related failures during simulation for reasons related to the integration method and the associated time-step control used by the simulator or iterative solution of nonlinear equations. These failures can be categorised as failure to compute a DC operating point and failure to find a solution due to time step reduction below a certain limit. (Vladimirescu 1994) describes the most common causes of convergence failures expected in SPICE based simulators and appropriate remedies. In brief, the typical cause of failures in transient simulations can be overcome by not using ideal component models and component models with default values. Relaxing the simulation settings such as resolution without compromising the simulation results in accuracy and increasing the iteration level helps achieve simulation completeness. As will be seen in Chapters 3 and 6 simulation settings were relaxed to allow a voltage and current waveform accuracy level of 1 mV and 1 mA respectively in contrast to the default level of 1 μV and 1 pA for voltage and current respectively. This is adequate for the power electronics circuits dealt with in this work because the voltage and current level present in these circuits and devices are of the tens of millivolts and milliamps level. A recently published paper (Parthipan et al. 2013) documents some of the literature related to this work based on the theory published in (Vladimirescu 1994) for SPICE3 based simulators.

The small time step requirements are not always met by the simulation tools and this leads to non-convergence issues during transient simulation. SPICE has built-in algorithm (Newton-Raphson (Shilpa & Taranjit 2003)) that automatically adjusts timing to adapt to an appropriate length time step. However, circuits do not always converge due to nonlinearities, discontinuous switching, high rate of change of current (*di/dt*) and high rate of change of voltage (*dv/dt*). The systems concerned are considered to be low-bandwidth. This means the switching

speed of the components and the transients is not expected to exceed 300 kHz. Moreover,, as will be seen later in Chapters 3,4 and 6, the circuit models of the LRUT pulser-receiver and the EBW power source require to be simulated for 100s of milliseconds before a usable results can be obtained. To add to the complexity, signal variants with different amplitudes and frequency ranges are likely to be present in the circuits. Work published by (Pedram & Wu. 1999) and (Uwe et al. n.d.) on time base simulation issues, suggests that to numerically analyse circuits, the numerical time step has to be at least 100 times smaller than the period of the maximum frequency (< 1/(100 x Maximum frequency)), making the smallest time step in the region of 30 ns for the analysis of 300 kHz signal in this research work. This greatly extends the simulation computation time, sometimes many hours are required just to get a single waveform and display the waveform from the disk. Transients due to high *di/dt* and *dv/dt*, make simulation even longer as even smaller time steps are required. High computing power and fast processing speed are essential for finishing the simulation and analysis in the same time phase.

SPICE3 based simulators PSpice and LTSpice handle non-convergence issues by allowing the user to run the simulation with a number of different options for solving linear equations and nonlinear solutions (Parthipan et al. 2013) (Vladimirescu 1994). Discussion on the algorithms implemented in the simulation tools is outside the scope of this research work.

1.6.5 System modelling methodology

The success of system modelling and simulation depends on the accuracy of the models used in the simulation. Computer simulation results will never fully map reality. However, the EDA based virtual test platform has a number of benefits such as no need for hardware, parameter optimisation analysis, tolerance test and individual component performance test which are not possible with real hardware.

Systems can be modelled using bottom-up and/or top-down methods. Bottom-Up methodology requires models which map the real elements that form the components and/or subsystems, hence topology specific. The estimation and data derived from this type of modelling are more accurate and map real hardware data. Several specifications and functions can be accurately simulated using the models, which are built using this method. Design schematics can also be derived or extracted from the modelling template. Since it is topology specific, the options of evaluating and estimating other topologies are limited. The modelling is more challenging as information on device specific physical parameters are required. Top-Down Methodology is topology free and can be generalised to be used with various topology options. This method is based on estimation using analytical expressions.

The systems that underwent modelling consists of multi-engineering discipline characteristics because of the inter-coupling nature between domains. This often cannot be modelled following a topology specific route. Moreover, the complex algorithms used in topology specific models slow the simulation and pose convergence issues. Besides, component manufactures do not always provide topology specific models. It is almost impossible to write topology specific models as the component manufactures do not publish the required sensitive data. For these reasons, in system model development, mixtures of topology specific and topology free models are often used.

1.6.6 Meeting simulation software requirement

Satisfying all the simulation requirements is technically and financially not feasible. Hence flexibility and work around methods need to be explored for the selection of a tool and also in the modelling methods. The major obstacles found were twofold: one, the software tool has to meet the requirement of modelling multi-engineering domains in a single simulation

platform with minimal or no financial burden. Simultaneously simulating the coupling nature between engineering-domain in a single platform is vital for achieving an acceptable level of confidence. The second issue was the convergence problems due to high *di/dt* and *dv/dt* during time-domain (transient) simulations. The fast switching of the non-linear switching devices can push the minimum time step requirement to the nanosecond (ns) range to converge. This also extends the simulation time to hours, if not days.

Chapter 2 and 5 highlight the complexity of the pulser-receiver and the EBW power source respectively. In brief, modelling the inter-coupling nature of the piezoelectric transducers and the battery, which operate in electro-mechanical and electro-chemical domains respectively and simultaneously simulating them in a single platform are important for the pulser-receiver power performance analysis. On the other hand, the EBW power source, which can be considered as a homogeneous electrical domain system requires models that can converge easily, during its high *di/dt* and high *dv/dt* operations. This system is a hybrid of non-linear power circuits, which during fast transient analysis cause non-convergence issues. Hence, it requires appropriate adjustments in the models for fast and accurate simulations.

There are numerous research works published on modelling the inter-coupling nature of foreign domain and simulating them in an electrical domain such as, PSpice (Michael 1998), (BS EN 50324-1 2002), (Jo de Silva et al. 2008), (Ramos, San Emeterio & Sans 2000), (Lauwers & Gielen 2002) (Benini et al. 2000), (Panigrahi T et al. 2001), (Rakhmatov & Vrudhula 2001), (Kroeze & Krein 2008), (Min & Rincon-Mora 2006) and (Gold 1997) where models were written in SPICE. Both systems, as will become obvious in the following chapters, fall into the category of electronic equipment/system with associated foreign domain components. Hence it is

wise if the simulations can be carried out in an electrical domain, with the inclusion of equivalent electrical circuit models of the other domain components.

Non-convergence issues can be overcome by either using simplified models and/or by relaxing the simulating setting. The choice of simplification on a model depends on application and availability of models and the required accuracy. The research work requires time-domain (transient) simulations for analysing functional performance of the systems.

1.7 Contribution to knowledge

The research focussed on two industrial applications (LRUT pulser-receiver and the EBW power source) which required reliable test platforms on which the systems performance can be analysed, so that necessary modifications can be implemented for future enhancements. The specific contributions to knowledge that had been made as an outcome of this research work are listed below;

- Development of a pulser-receiver model for power performance and functional analysis (Parthipan et al. 2011):

An application specific simulation tool box was built according to the specific need of power analysis for enhancing portability of an energy aware system that constitutes multi-engineering discipline components. The tool set was based on a licence free electrical domain simulation tool, but efforts were made to infer other engineering discipline constructs and simultaneously simulate the system model to analyse mutual coupling between the constructs. This virtual test platform contributes to rapid prototyping and evaluation of new conceptual designs.

- Development of new hardware for novel medium frequency LRUT application:

This work had resulted in evaluating and demonstrating a novel concept of applying LRUT technique to complex structures (Haig & Stavrou 2012). This hardware development process used previously developed simulation models to verify and evaluate the new circuits. This work demonstrated tool-based reuse and modification of models for automating and accelerating the design process. The systematic methodology followed in the previous model development work made almost all sub circuits models to be reused in this tool based process. Work allowed the industry to demonstrate medium frequency LRUT concept for the first time in TWI history.

- Model development of electron beam welding power source for hardware assessment (Parthipan et al. 2013):

The developed system model represents multiple levels of abstraction in the EBW power source. In spite of this model's complexity, it allowed non-convergent free transient and AC simulations to be carried out and produce sensible simulation data that closely match reality in a realistic simulation time. The model made it possible to look at the interactions between the power stage, the control electronics and the load for a component level assessment of this system during fault and various operational conditions. Reproduction of fault conditions and parameterisation of component values for better assessment of hardware was also made possible with this developed model, which is almost impossible with real hardware. The outcome of this work also includes an equivalent circuit model of an off the shelf IGBT device DIM800DDM12-A000 that easily converge in circuit simulations while producing simulation results that closely match reality.

- Fault recovery and system stability analysis of the electron beam welding system:

Simulation of this developed model made it easy to determine and/or reassure theoretically derived optimum control parameters for the circuits that govern the fault recovery sequence and the system feedback control. Developed model allowed quantitatively measure each and every node and constituent component behaviour during and aftermath of a simulated fault condition – flashover by simulation; a carefree procedure where the risk of damaging the hardware and human error were completely avoided/reduced. Likewise simulation of this developed model allowed better understanding the system's stability and the major factors that influence the stability of this system, with the outcome of optimum component values for the compensation network that best serve this system. Identification of the optimum parameters theoretically and evaluating them in a simulation gives the confidence for implementing this in the real hardware for welding trials.

1.8 Structure of the thesis

This Chapter introduced the scope of the thesis, concept of system modelling and aspects related to the modelling the two concerned systems effectively for reliable assessment. Chapter 2 provides an introduction and background information about the principle of operation of LRUT technique and the electronic system that facilitates the technique. It also identifies the need for the enhancement and the sensitive components that require optimisation for achieving portability. Chapter 3 is the technical work carried out in the development of an adequate model for the LRUT pulser-receiver and associated tools for the assessing portability enhancement. The possibility of adapting existing pulser-receiver hardware for the enhanced LRUT technique for inspection of complex structures are analysed in Chapter 4.

The EBW concept and the hardware that makes this industrial application possible is introduced together with a discussion on an application specific issue of flashover in Chapter 5. This Chapter also identifies the critical constituent components that required modelling and simulating for the assessment of the EBW hardware for application optimisation. Chapter 6 discusses the development and the evaluation of the EBW power source simulation model with case studies of utilisation of the developed model. Chapter 7 concludes this thesis with the discussions on further work. Appendices A and B include some of the main models discussed in Chapters 3 and 6 respectively.

1.9 Publications

- Parthipan, T., Nilavalan, R., Mudge, P., & Wamadeva, B. (2010). Ultrasonic pulser echo system modelling. *Proceedings of the 2010 Conference for the Engineering Doctorate Programme in Environmental Technology*. London.
- Parthipan, T., Mudge, P., Rajagopal, N., & Wamadeva, B. (2011). Design and Analysis of Ultrasonic NDT Instrumentation Through System Modelling. *International Journal of Modern Engineering, 12*(1), 88 – 97.
- Parthipan, T., Nilavalan, R., Mudge, P., & Wamadeva, B. (2011). Power estimation by system modelling for reliable structural integrity modelling. *Proceedings of the 2011 Conference for the Engineering Doctorate Programmes in Environmental Technology & Sustainability for Engineering & Energy Systems.* Surrey.
- Parthipan, T., Ribton, C., Mudge, P., Rajagopal, N., & Wamadeva, B. (2013). Enhancement of high voltage Electron Beam Welding power supply: Rapid recovery

after flashover detection for void-free welding. *Industrial Electronics (ISIE), 2013 IEEE International Symposium on.* Taipei.

2 LONG RANGE ULTRASONIC TESTING

2.1 Overview

This Chapter describes the concept of Long Range Ultrasonic Testing (LRUT) that is used for non-destructively testing industrial components for defects using ultrasonic waves. An introduction to the electronic equipment and associated components that facilitate implementation of a typical LRUT technique is also given. The Chapter also discusses the need for the enhancement of the existing system and the utilisation of electronic system modelling techniques to achieve this. An enhanced LRUT technique – medium-frequency LRUT, is also introduced followed by an introduction to the extension of research work, where the developed system model for enhancing the typical LRUT system was utilised for realising hardware for medium-frequency LRUT.

2.2 LRUT technique

Ultrasonic Non Destructive Testing (UT) is a method of inspecting the material condition of structures and components using ultrasonic waves (Mudge 2006). This method ensures that the material and its mechanical properties are not damaged or changed during inspection. LRUT is a subset of UT that has the ability to detect defects for many metres in different shapes and types of structures such as pipes and plates. This technique enables the inspection to be carried out from a single point of access if required. In contrast to conventional UT this novel technique utilises the lower end of the ultrasonic spectrum in the frequency range of 20 kHz to 100 kHz in typical applications.

2.3 Plant Integrity's LRUT system

Plant Integrity Ltd. (PI Ltd.) specialises in the design, manufacture and marketing of instrumentation that facilitate the LRUT technique. A battery powered electronic equipment - pulser-receiver unit facilitates the LRUT technique to be applied on structures in the field. The unit uses ultrasonic waves to detect anomalies by sending ultrasonic waves along the specimen and then processes the ultrasonic waves received. The received signal can be either responses from features or defects when operated in the pulse-echo method or the shadowed image after the defects in the pitch-catch method (Gharaibeh 2008), (Mudge 2006).

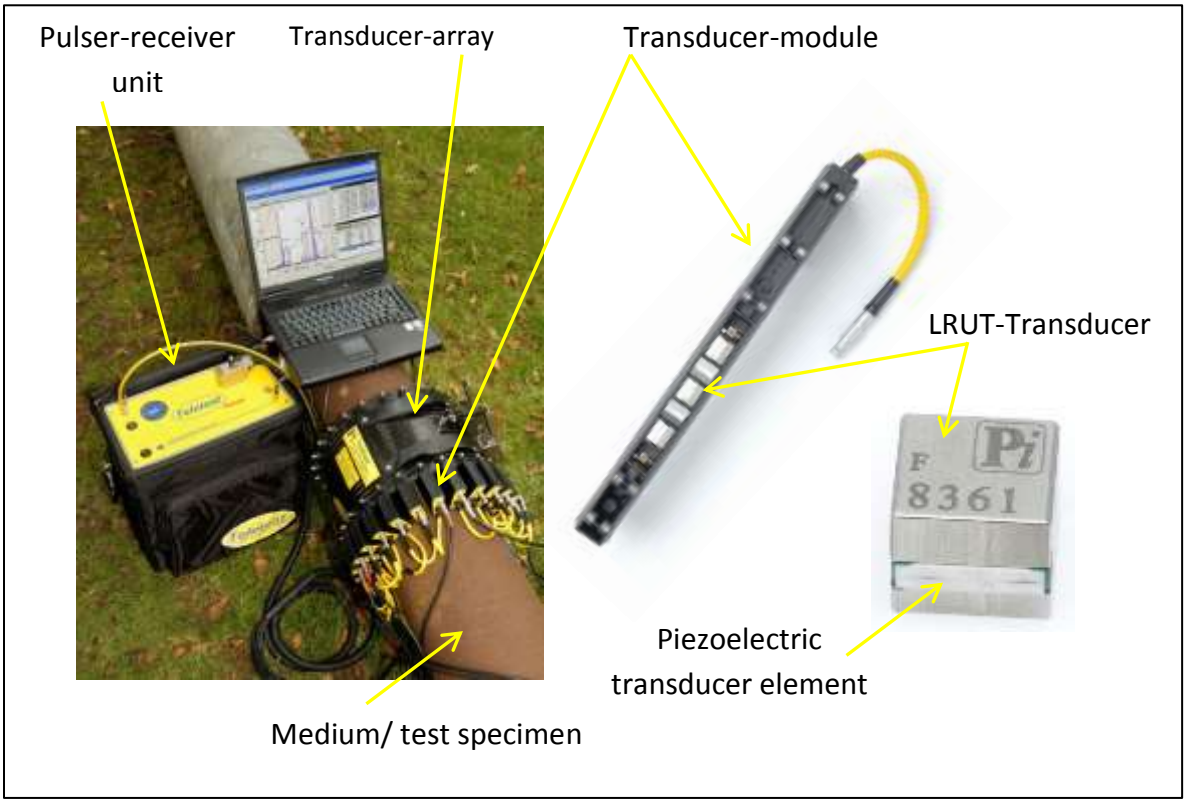


Figure 2-1 Example LRUT system and its components – PI Ltd.'s system

The pulse-echo method is a way of transmitting ultrasonic waves from one location and receiving the echoed energy from the same location using the transmitting sensor as a receiving

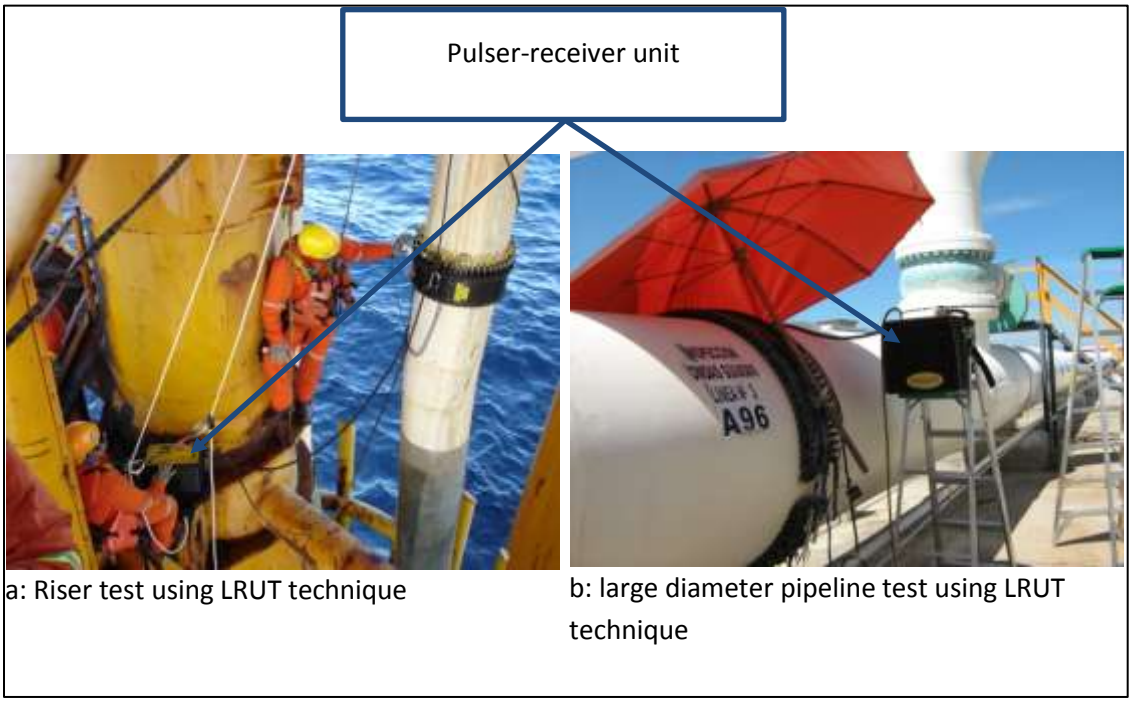
sensor. In the pitch-catch mode an ultrasonic wave is transmitted from one location and the shadowed energy is received from a different location using different sensors.

The system can be explained with the aid of Figure 2-1. The pulser-receiver unit uses LRUT-transducers as transmitters and receivers. The LRUT-transducer is a pre-engineered transducer assembly that uses piezoelectric transducer (PZT) elements of type EBL#2 (EBL Piezoelectric precision 2010). The PZT element produces ultrasonic waves when excited with a varying high voltage, short duration electrical signal. This element is also capable of producing electrical signals at its electrical terminal when its mechanical terminals (body) are stressed with a mechanical force – i.e. an ultrasonic wave (Jose, San & Antonio 2004), (BS EN 50324-1 2002). A number of these LRUT-transducers are pre-arranged in the transducer-array – also known as a collar, by means of transducer-modules to generate preferred wave-modes in the medium when excited. The medium is the test specimen. Acoustic coupling is achieved by dry-coupling. The collar is inflatable and in typical applications, the collar is wrapped around the specimen and inflated to the maximum of 60 psi (~ 4 bar) for achieving adequate acoustic coupling between the LRUT-transducers and the test specimen. An external personal computer or laptop computer (PC) is linked to the pulser-receiver unit to configure the equipment and to process the received data. The PC is installed with proprietary software that governs the LRUT process.

2.4 Applications

The ability of the LRUT technique to test pipelines in service, meaning there is no need to shut down/stop production for inspection, has been attractive to many industrial sectors. This is especially the case for oil and gas industries with a need to assure operational fitness of their

facilities. These facilities are on-shore and off-shore facilities mostly in remote locations. Two features in such facilities; risers in offshore platforms and large diameter pipelines that underwent inspection using typical LRUT technique are shown in Figure 2-2; a and b respectively to illustrate the extreme nature of the environment where LRUT instrumentation is used. Portability and reliability of the instrument is paramount for the ease of usage and installation.



a: Riser test using LRUT technique

b: large diameter pipeline test using LRUT technique

Figure 2-2 LRUT of offshore risers and remote pipelines using a pulser-receiver unit

2.5 Application specific issue – portability

The research reviewed the existing equipment to identify areas that could be improved from a portability point of view. The existing pulser-receiver unit (referred to as MK3) is less than ideal for portability as it weighs about 16 kg and measures 400 mm x 400 mm x 300 mm. It is equipped with a 30 V; 13 Ah rechargeable Lithium-Ion (Li-Ion) battery as a primary power source. The 30 V voltage level is firstly down-converted to 12 V using commercial DC-DC

converter modules and then up and down converted using a number of custom made switch mode power supplies (SMPS) to all required voltage levels in this system. These power supply circuits were identified as inefficient and bulky due to the usage of heat exchangers (heat-sink and fans) and caused EMI issues, preventing the unit being EMC certified.

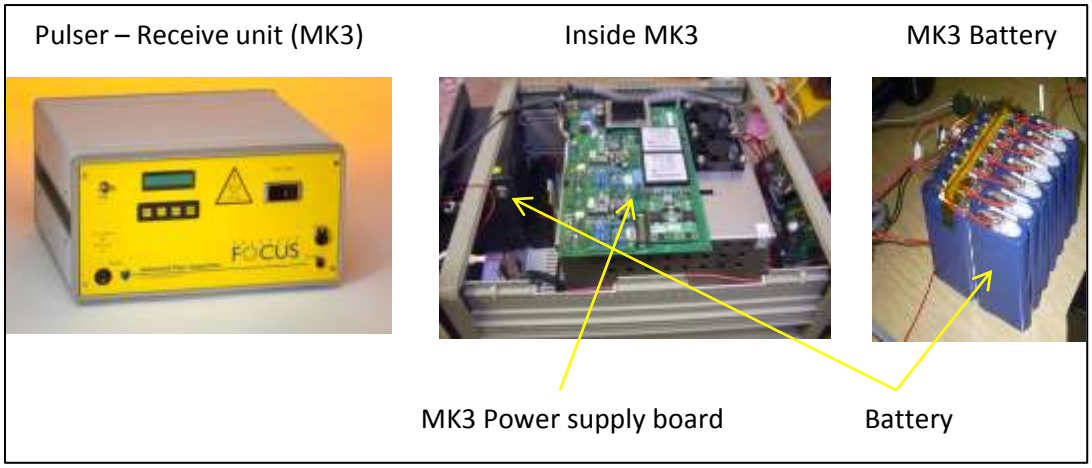


Figure 2-3 MK3 Pulser-receiver unit and its internal view

The capacity of the battery for the application is assumed to be over specified in terms of power density (W/ kg) and volumetric energy density (Joules/cm^3) for the portable application this instrumentation is intended to do. A fully charged battery utilised in the system lasts just under 72 hours, whereas the expected operational time of the system is only 12 hours due to operational constraints. Figure 2-3, shows a picture of the MK3 version of the pulser-receiver unit and an internal view illustrating the space occupied by the battery and power supply PCB.

The LRUT application is a pulse-load application, which requires dumping of high energy in the range of 6 Joules (pulsed at peak voltage and current level of 120V and 1.8A respectively on each one of the 24 channels) for 1 ms at a pulse repetition rate of 0.1 s to the load, while maintaining the load excitation voltage. This was facilitated by a ± 150 V DC high voltage SMPS, known as HTPSU, with a bulk capacitor bank of capacitance value 3200 µF

(rated at 400V) in total. The physical size of the bulk capacitor bank (HT-Bank) was assumed to be over specified for the application, as the presumed energy dump requirement of 6 Joules in principle does not require an HT-Bank of value more than 1000 μF in total. This is based on delivering energy of 6 J, while maintaining a tolerable voltage drop at the HTPSU's output rails and this tolerable voltage drop defines the capacitor value. The capacitor equation for energy $E = CV^2/2$ was used to calculate the bulk capacitor values for +150V and -150V rails individually. As will be seen in Chapter 3, maintaining these supply rails within an acceptable ripple level is crucial for adapting to the Power Supply Rejection Ratio (PSRR) of the drive circuits that excites the LRUT-transducer/s. So the possibility of reducing its physical size credited further review.

2.6 Limitations

Addressing the application specific issue of optimising the battery size, power supply circuits enhancement/prototyping and the reduction of the HT-Bank size without compromising the operational performance of the pulser-receiver required careful attention. Reliable operation of the pulser-receiver unit for the scheduled inspection period could have been jeopardised if the system was not adequately powered (AboElFotoh, Iyengar & Chakrabarty 2005), (Wei 2007). Moreover power requirements can vary from scenario to scenario and function to function (Luo & Jha 2001). Hence knowing the power requirement of each and every scenario and function that the LRUT system intends to do is important. The power requirements and usage between charges of the system set a constraint on the minimum battery capacity. The power requirement also depends on constraints external to the system such as temperature, energy source supply variations, load variants, human error and system rest time. Reliably predicting the power budget of the instrumentation for various configuration and functionality is difficult especially at the

conceptual stages of the design. Even when the hardware has been built, parameterising every single function is resource intensive and consequently may well be overlooked.

2.7 Research methodology

The research implements a system modelling technique to analyse the possibility of optimising the pulser-receiver unit. This technique allows systems to be studied for obtaining enhancement values/parameters with minimal or no hardware prototyping requirement. The main components whose functionality is likely to be affected by the downsizing of the battery and the HT-Bank were first identified. Having identified these critical components or subsystems, they were modelled and simulated in the electrical domain, using the simulation and modelling tool LTSpice for analysing their transient behaviour. These simulations specifically concentrated on the power performance and the peak current usage for optimum LRUT inspection parameters in order to derive optimum specifications for the battery and HT-Bank. Note that downsizing of the battery in effect reduces the terminal voltage and its capacity. Transient simulations were also used to evaluate the prototype designs of the power supply circuits and selecting appropriate topologies and components. This work is covered in Chapter 3 in detail.

2.8 LRUT system architecture

Understanding the LRUT system architecture helps reliable assessment of power and functional performance and subsequently enables enhancing it for optimised size, functionality and power performance. Figure 2-4, shows the main components of the LRUT system and it is used here as an aide to illustrate the LRUT system architecture. The pulser-receiver unit is an electronic system comprising a number of transmit and receive channels. Each transmit and receive channel is connected to single or multiple LRUT-transducers using wired transducer

modules mounted in the transducer array. The transducer arrays for pitching (transmitting) and capturing (receiving) the ultrasonic wave are labelled as $TX_{TRANSDUCER\_ARRAY}$ and $RX_{TRANSDUCER\_ARRAY}$ respectively. The picture depicts the pitch-catch configuration. In pulse-echo mode, $TX_{TRANSDUCER\_ARRAY}$ also acts as $RX_{TRANSDUCER\_ARRAY}$. The pulser-receiver unit is the only component in the LRUT system that requires power to operate. Consequently the power performance analysis is dependent upon understanding the pulser-receiver unit's construction and the factors that affect/influence its power performance. The PC is not included in the study as it is powered independently.

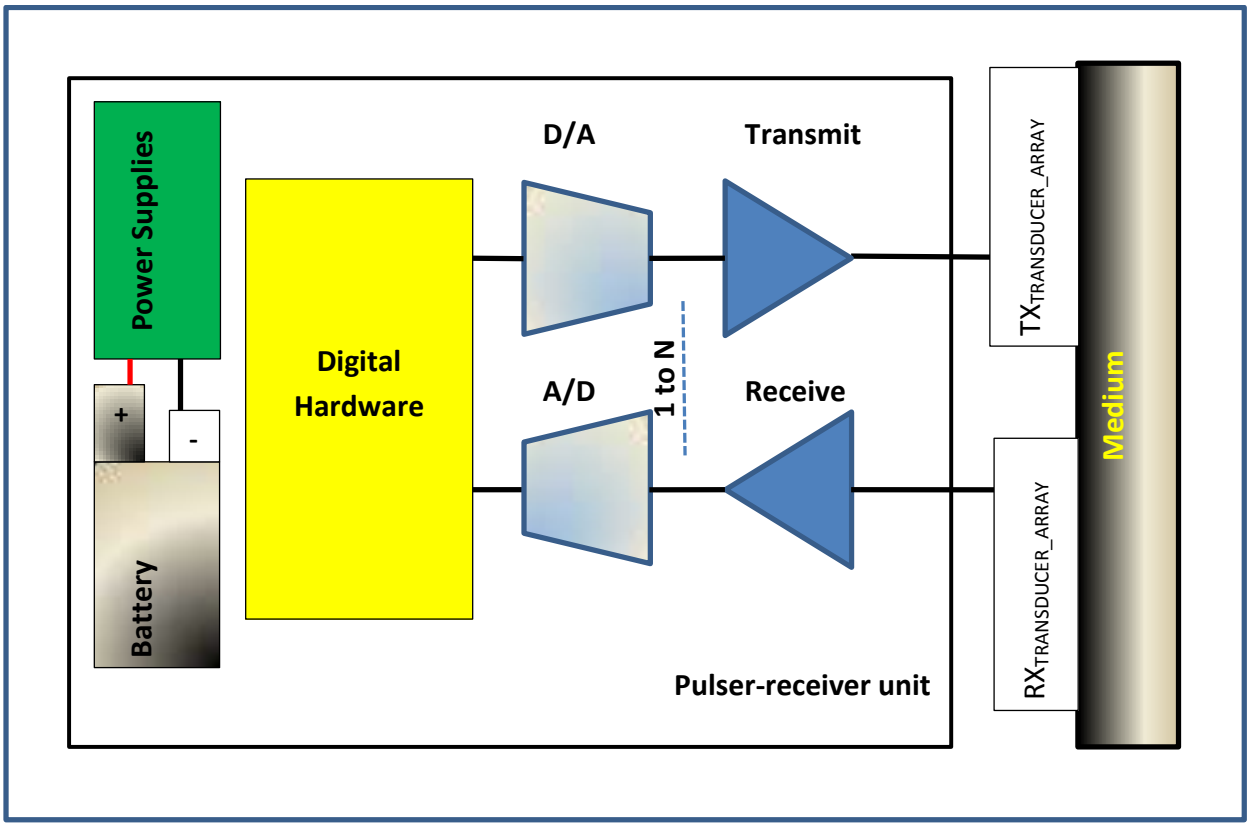


Figure 2-4 Simplified architecture of LRUT system

The following subsections discuss the functionality of the pulser-receiver unit followed by the main components/ sub systems within the system and the impact on their sub-functionality by the size reduction of the primary power source, the battery and the HT-Bank.

2.8.1 Functionality

The functionality of the pulser-receiver unit can be expressed as follows: the excitation signal profile (excitation profile) with respect to its frequency, amplitude and the phase delay between transmit channels and the pulse repetition rate (PRR) are all important for achieving the preferred wave mode in the test specimen. This excitation profile is generated as per user settings by TWI proprietary software, installed on a PC and downloaded into the equipment's on-board memory. This download activity also configures the pulser-receiver unit's hardware to a specific test requirement. Configuration settings include receiver gain settings and filter settings. The mode of operation (pulse-echo or pitch-catch) is also defined at this stage. A collection (inspection activity) is demanded following the configuration of the hardware, which forces the downloaded excitation profile to be converted into its corresponding analogue low voltage excitation waveform by a digital-to-analogue converter (DAC). This waveform is subsequently amplified to a high voltage excitation signal (approximately to a voltage level of 240 V AC) by the transmit circuits (drive circuits). The transmit circuits also function as a current source to deliver a high load current of approximately 3 A per transmit channel in heavy load conditions. The transmit circuit output subsequently excites the LRUT transducers (in transmit mode) that are connected to each transmit circuit output. The excited transducers produce an acoustic wave which propagates in the test specimen.

In the pulse-echo mode once the transmit function is completed the unit switches to receive mode, in which the transmit transducers are isolated from the transmit circuit/channels and are connected to the receive circuits, making the unit ready for capturing the energy reflected back from features (e.g. defects and edges) of the test specimen. In pitch-catch mode the receive transducers are connected to the receive circuits during the entire cycle as the transmission is handled by separate transducer arrays. The receive circuits condition the analogue received/captured signal, before digitising it, using an analogue-to-digital converter (ADC). The digitised data is then stored in an on-board memory (after averaging for noise reduction), before being transferred to the PC via a communication link.

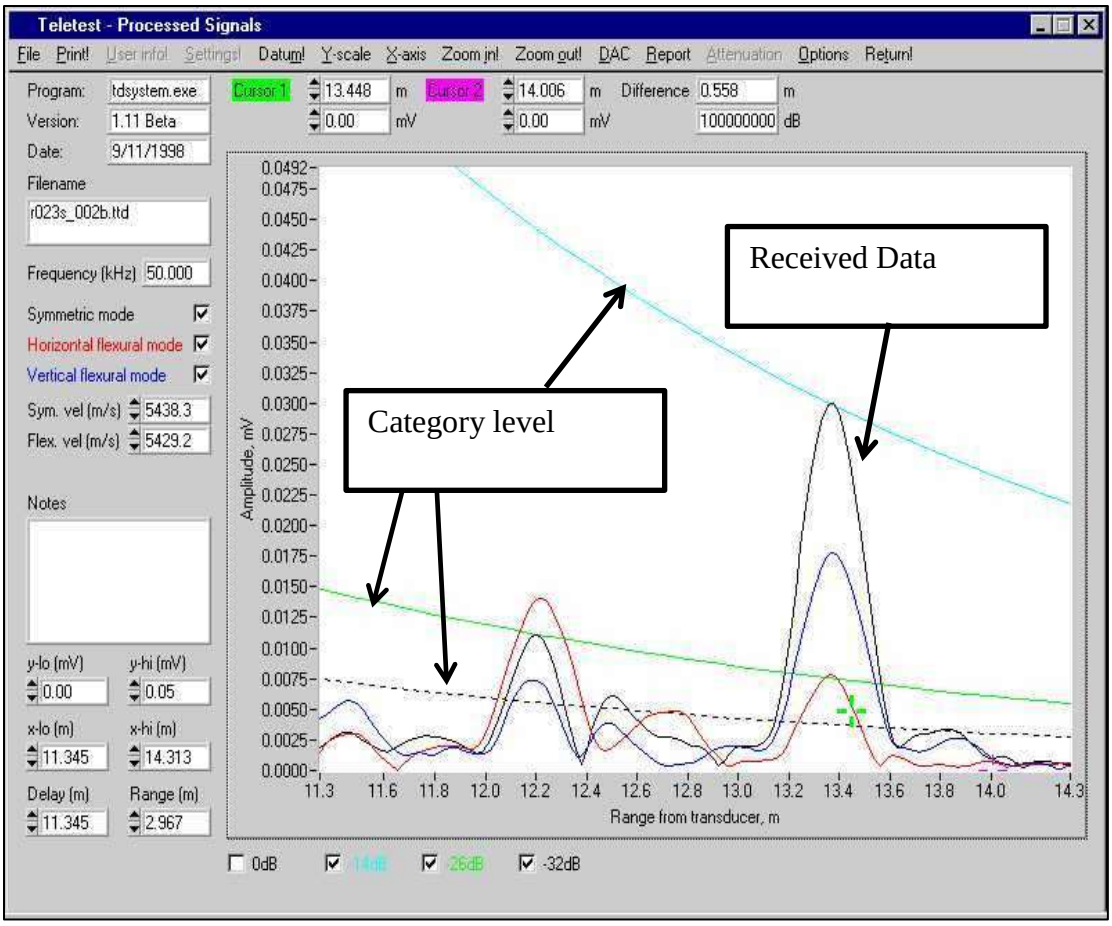


Figure 2-5 A-Scan showing category level guide superimposed on captured data

The transferred data is then signal processed by the TWI proprietary software in the PC for displaying the A-Scan. An A-Scan is a format of graph that displays the received signal amplitude on the y-axis and distance of features from the datum point along the x-axis. The datum is usually the location where the transducer array is mounted or a known feature (such as a weld) on the test specimen (selectable). The graph superimposes a category level guide on top of the displayed data as shown in Figure 2-5, to assist decision making on potential anomalies.

2.8.2 Load

The load to pulser-receiver unit is a number of LRUT-transducers that are pre-arranged in a planned fashion to form a transducer-array. A variation in load affects the dynamic performance of the transmit circuit and consequently affects the power performance of the pulser-receiver unit. This variation can be because a number of LRUT-transducers require excitation and/or a change in excitation frequency or due to interconnection cables and damping effects caused by coupling. Hence having a fair knowledge of the load behaviour is important for reliably analysing the system's power performance.

The LRUT-transducer is a transducer assembly, each element of which consists of a piezoelectric transducer, a damper and face plate. Detailed discussion on its construction is covered in Chapter 3 and in appendix A. The piezoelectric transducer is an electro-mechanical transducer and is capacitive in nature at its electrical terminal, when excited below its series resonance frequency. Its electrical terminals are excited and sensed by the pulser-receiver unit in transmit mode by the transmit circuit and receive mode by the receive circuit respectively. These transducers require high voltage AC or a pulsed-DC voltage stress to force them into oscillation. A typical LRUT application uses high frequency excitation voltages, whose frequency can vary

from 20 kHz to 100 kHz in order to achieve adequate resolution. The varying frequency changes the impedance of the capacitive load, making the loading effect lighter or heavier. The number of transducers connected to each transmit channel can vary from application to application. In receive mode the LRUT-transducers are stressed mechanically, and for the pulser-receiver unit, these transducers are passive, i.e. no extra power is used to make the transducer sense the received acoustic/mechanical signals. However, the receive sensitivity of the LRUT transducer plays a part in the signal conditioning of the received signal.

The HT-Bank is the power supply bulk capacitor for the transmit circuit that drives the LRUT-transducers. In a typical pulser-receiver unit there are 24 transmitting channels demanding approximately 72 A in total for the duration of 1 ms at a PRR of 0.1 s. to sufficiently excite the heaviest load. The heaviest load is when each transmitting channel supports 16 LRUT transducers connected in parallel and the excitation signal is of a frequency of 100 kHz and peak-to-peak amplitude 240 V.

The reduction of HT-Bank capacitance, hence the reduced energy stored in it, may not be sufficient to meet the heavy load demand or may not be sufficient for the HTPSU to maintain the HTPSU supply rails ripple-free for satisfying transmit circuit PSRR requirements. In other words, the residual charge left in the HT-Bank after each excitation may be too low for the HTPSU to recharge the HT-Bank to regulate the supply rails. It is crucial that the HT-Bank tolerates the surge current required at the beginning of the excitation as well, caused by the short circuit nature of the capacitive load. This scenario is documented in Chapter 3; section 3.3.2.

The research was also looking to both reduce the capacity and terminal voltage of the existing battery. If a high current is drawn by a heavy load, it can drain the battery quicker than

the planned operational period. Moreover, the effect of a lower battery terminal voltage makes the HTPSU work harder and possibly makes it inefficient as it has to deal with a higher rate of change of current (the primary current would be higher to satisfy the load demand). Hence, use of a lower capacity battery may lead to a requirement for higher rated components for the HTPSU, which leads to a higher cost and increased physical size of the HTPSU, which counteracts the goal of achieving portability.

2.8.3 Pulser-receiver unit

This entity is a construct of analogue, digital and mixed signal components. Its architecture has a number of low level subsystems as shown in Figure 2-4. These subsystems are briefly discussed here and dealt with in more detail in Chapter 3. The power related performance of these circuits is influenced by the current and voltage waveforms these circuits are working for different operating conditions and various functions.

2.8.4 Digital hardware

Digital hardware in the pulser-receiver unit includes control logic, data-storage and the communication circuit mainly. This is based on CMOS technology and in general is supplied with a well regulated low voltage DC SMPS. Its functionality is unaffected by the HT-Bank directly, but the downsizing of the battery has an effect on its operational duration. Hence having a knowledge on this subsystem's power performance is beneficial for adequately specifying the battery capacity. In general, the supply voltage and system clock frequency affect the power consumption of the CMOS digital logic. Digital logic includes the following main components and/or subsystems:

- Field Programmable Gate Array (FPGA): An FPGA is an integrated circuit that can be configured by the user after fabrication. The programming data is written using a hardware description language e.g. VHDL or Verilog, which configures the hardware of the gate array. The pulser-receiver unit utilizes one of these devices for control, housekeeping and data manipulation purposes. The device is a digital hybrid with CMOS input/output ports (banks). Its functionality and the number of gates used in the FPGA (resource usage) is very well defined during the programming/configuration stage of the FPGA. Hence, its power consumption can be reliably predicted using vendor specific software if the circuit design netlist is available. The netlist is a gate level description of the hardware, generated by synthesising the VHDL/Verilog hardware description. This has all the physical information of the gates. However, it is best if the worst case scenario is considered when budgeting for power performance. For the FPGA device used in the design, when 100% of its resources are used and toggled at the maximum clock frequency it is likely to consume in the region of 10 W worst-case (Xilinx Spartan-3A DSP 2011). More information on this is included in Chapter 3; section 3.3.4.
- Memory Devices: The unit consists of two types of memories – DDR2 and asynchronous-SRAM which are considered power hungry. These devices are also based on CMOS technology. These are for temporarily storing the received data prior to transferring it to the PC. Memory device DDR2 is a Double-Data-Rate synchronous memory device that requires the presence of a clock for its operation. Asynchronous SRAMs are based on time stamped signals for data transfer. The power demand by these devices is the same every time their service is required and in general they consume 1 W power during activity per device.

- Communication peripherals: The equipment uses Ethernet 10/100-MBPS protocol for communicating with the PC. This protocol is handled by the Ethernet controller Intellectual Property (IP) core, integrated in the FPGA. Hence the FPGA power budget includes the communication circuit's power consumption data. However, the physical layer (PHY) for the Ethernet controller is external to the FPGA. As these PHY devices are made to IEEE 802.3 22.2.4.1.5 standard these devices feature a power down mode when not used and consume insignificant power i.e. 150 mW and 17 mW during active and power down mode respectively. The communication is active only when the excitation profile is downloaded and when the data is transferred from the unit to the PC. This process takes under 1 minute in typical operation.

2.8.5 Data converters

There are two types of data converters - ADCs and DACs used in the unit for interfacing between the analogue and digital domains in the electronic circuits. These devices are mixed signal components and consist of analogue and digital circuits. ADC converts a received analogue signal into a digital signal, so that it can be stored in the on-board memory. DAC converts the excitation signal stored in the on-board memory in a digital form into its corresponding analogue signal for it to be amplified before exciting the load. Their part in the power performance is very much influenced by the sampling frequency and the supply voltage they operate at. Due to their low supply voltage (2.5 V) and drive current capacity (tens of mA) their power consumption in this electronic design is considerably small. However, there are 24 ADCs and 24 DACs in the unit.

The optimisation of the HT-Bank does not impose any effect on the data converters. However, like the digital hardware their operational period can be jeopardised by the reduction of the battery capacity.

2.8.6 Transmit circuit

The transmit circuit is used to drive the piezoelectric transducer load. The manner the load is excited, requires the transmit circuit to be powered with high voltages and deliver significant amount of load current. This makes this subsystem a high power consuming and power dissipating construction. Another aspect associated with its operation is that, for its reliable and undistorted functionality, the high voltage power supply (HTPSU) to this circuit has to be stable during standby and when there is a high power demand during load excitation. The HT-Bank is responsible for holding sufficient energy for this and the consequences of an under engineered HT-Bank will be to distort the performance of the transmit circuit. Moreover, the load variation and the excitation profile are very much linked to the transmit circuit's dynamic performance. So understanding the inter-dependency between the transmit circuit, the load and the HT-Bank is crucial for best optimisation of the HT-Bank.

The downsizing of the battery in terms of its capacity will affect this circuit's operational duration and lowering the battery terminal voltage will make the HTPSU work harder to adequately regulate the high voltage supply to this sub circuit. There are 24 of these circuits in the unit and as will be seen in Chapter 3; section 3.3.1, a significant amount of power is consumed and dissipated by this sub system.

2.8.7 Receive circuit

The receive circuit accepts the electrical signal from the LRUT-transducers, when the pulser-receiver unit is in receive mode. It signal conditions the received signal so that the useful data can be digitised and stored in the memory without losing valuable information for data manipulation. Signal conditioning includes voltage level amplification and filtering out unwanted signals. This circuit is supplied with a well-regulated low voltage DC power supply and the stability of this supply is vital for its undistorted operation. Signal conditioning of the input signal requires different levels of amplification, depending on the attenuation level experienced during LRUT inspection. The receive circuit was designed to have configurable amplification levels that can be adjusted from 20 dB to 100 dB in 1 dB steps, prior to data collection. This is done with a number of analogue-switches, which are controlled by the FPGA, based on the configuration data downloaded in the configuration stage. The hardware filters designed in the circuit for filtering out the unwanted noise/signal are also configured in the same way.

The size reduction of the HT-Bank does not directly affect the receive circuit's performance, but the distortion it can impose on the transmitted ultrasonic wave can degrade the quality of the received signal. The downsizing of the battery capacity will jeopardise the operational time of the circuit, but the lower terminal voltage does not directly affect its performance as this circuit is provided with a regulated power supply. There are 24 of these circuits in the unit and due to the high component count - in excess of 3000 in this sub system, it is assumed to be one of the high power consuming subsystems in the unit when active.

2.8.8 Power supplies

There are a number of SMPSs power supplies used in the pulser-receiver unit to serve the purpose of DC-DC converting the battery terminal voltage to relevant voltage levels that are required by each sub systems and circuits in the unit. Providing adequately regulated power supplies is crucial for the correct functionality of all these circuits. SMPS were used for achieving a better efficiency and a small form-factor. In contrast to linear power supplies, SMPS allow a large difference in the input voltage level and output voltage level, while preserving a higher order efficiency (above 70% with careful design in almost all SMPS design topologies). The input voltage to these SMPS is provided by a rechargeable Li-Ion battery.

Downsizing the battery has two major consequences associated with this sub system in general. One, as the terminal voltage is reduced; the difference between the input and the output of each power supply in the sub system is large, which makes the SMPSs work with a higher switching frequency. This leads to an increase in power consumption and higher power dissipation by the SMPSs. Two, the current drawn from the battery would be higher to maintain the power delivery to the secondary side of the SMPSs. The effect of this can be to jeopardise the efficiency of the SMPSs and reduce the operational time due to a higher discharge rate from the already downsized capacity battery. Moreover, heat exchanging elements such as heat sinks may also be required, which are detrimental to portability.

The HT-Bank is directly linked to one of the SMPSs in this sub system; HTPSU that charges the HT-Bank. The HT-Bank is the load to this HTPSU and reducing HT-Bank's size would make the charging time shorter and consequently less work for the HTPSU during charge cycle. The other SMPSs in the sub system are not affected by the downsizing of the HT-Bank.

Details of all SMPSs implemented in the new prototype are documented in (Parthipan 2010). In brief, a number of SMPS topologies, such as single-switch and two-switch buck converters, current-mode synchronous buck converters and current-mode flyback converters are used to achieve all the required power domains in the newly prototyped power supplies. Careful design practice together with the usage of performance optimised power products achieved an acceptable level of efficiency 86%, in fact closer to the achievable maximum efficiency published in manufacturers' datasheets in all cases. This thesis includes detailed work on the HTPSU design in Chapter 3; section 3.4.1 as it influenced the optimisation of the pulser-receiver unit the most and demonstrates the research methodology best. The work carried out on other SMPSs is not repeated, but the achieved efficiency on each SMPS topology is included in Table 2-1 for completion.

Power domain	Topology	Efficiency achieved
+ 5V @ 12 A continuous	buck DC-DC conversion using two-switch forward topology	88.5 %
-5V @ 2 A continuous	buck DC-DC conversion using single switch forward topology	84.5 %
+12V @ 12 A continuous	Current mode synchronous buck converter	98.4 % (Heavy load); 70.3 % light load)
+24V @ 1 A continuous	buck-boost DC-DC converter that uses two-switch forward topology	98.5 %
-24V @ 1 A continuous	current mode positive to negative DC-DC conversion using flyback topology	95.5 %
± 150V @ 72 A pulsed	flyback topology	~ 80 %

Table 2-1 Topologies used to construct the pulser-receiver unit power supplies

2.8.9 Primary power source – the battery

The pulser-receiver unit uses an electrochemical rechargeable Lithium Ion (Li-Ion) battery as its portable energy source. Downsizing the battery would affect two aspects of the battery; the terminal voltage and the capacity. Maintaining the terminal voltage above circuits

and sub systems minimum operational voltages is crucial for keeping the circuits operational. As the battery capacity drops due to discharge, the terminal voltage also drops. In energy aware systems, like the pulser-receiver unit, the number of complete functional cycles the system can perform, before its battery terminal voltage drops below the minimum operating voltage is quantified for making sure the system stays operational for the scheduled operational period. As will be discussed in Chapter 3; section 3.5 the optimised pulser-receiver unit for size and weight is expected to carry out 12 complete LRUT inspections before the battery requires recharging. Achieving this is in essence the ultimate goal of this research.

2.9 Identification of crucial components

The following subsystems have been identified as crucial components that require modelling and simulation for power performance analysis and subsequently in the decision making for optimising the pulser-receiver unit for enhanced portability.

- Load:

The battery and the HT-Bank sizing are very much linked to the load effect. With respect to power performance and the loading effect on the pulser-receiver unit, it is the input impedance of the LRUT transducer array that influences the most. In this case, the input impedance is determined from the AC characteristics of the LRUT-transducer or the transducer-array represented by the Voltage/Current ratio, measured across its electrical terminals.

The input impedance value of the piezoelectric transducer changes not only due to the varying excitation frequency, but also by the assembly of the backing block and possibly by the acoustic coupling effect influenced by the test specimen. This peculiar effect is a resultant of the

acoustic impedance imposed by the aforementioned elements on the mechanical face of the piezoelectric transducer. A detailed description of this effect is discussed in Chapter 3; section 3.2.2.

An impedance analyser can be used to study this effect, but it is difficult to distinguish the impact the load variation makes on the dynamic performance of the transmit circuit in operation due to so many variants in the system and practicality issues in separating the transmit circuit from the rest of the system. Modelling the piezoelectric transducer and the backing block and simulating these electro-mechanical assemblies with the transmit circuit allows the system to be analysed at a component level. It also allows parameterising concerned design specific parameters and component values, which is almost impractical in hardware tests. In addition modelling and simulating the transducer with the transmit and receive circuit together with the simulation model of the medium allows the port dynamics of the system to be studied. Chapter 4 specifically addresses this aspect.

- Transmit circuit:

The reasons identified for modelling the load are similarly valid for the transmit circuit. Its dynamic performance is coupled to the load variation and the HT-Bank size. Moreover, other factors such as port dynamics that include the excitation profile, required amplification of the excitation signal and the load demand also influence its dynamic performance. Due to the high voltage and high current performance the stress on its internal components is high and abrupt. On top of this, degraded supply rails due to an under-engineered HT-Bank, high load current and out of range port behaviour can contribute to oscillations and non-linear behaviour of the circuit components. Hardware measurements and theoretical calculations together can be used to assess

the impact on this subsystem, due to the intended changes in the HT_Bank and battery size, but this is only possible at a subsystem level and access to a component level assessment would be very limited. Component level stress analysis with parameterisation is easier and flexible with modelling of the subsystem and simulating it with the other components or subsystems whose functionality couples with this assembly.

- Receive circuit:

The receive circuits, regardless of their passive sensing transducers; consume a significant amount of energy due to their high component count. In addition to this, should the input (receive) signal level exceed the port dynamic range; this circuit would distort the useful signal and lead to an undesirably high power consumption. Analysing this is possible with measurement of prototype hardware, but at a cost and resource usage. The circuit is built-in with a number of level shifting and offset adjustment circuits, which require precise calibration for functional correctness. This is of course possible with careful design calculations, but accounting for collective tolerances of the components and parameterisation of component values is complex with calculation, but easier to deal with using simulation.

- Power supplies:

The necessity of producing new power supply subsystem for the pulser-receiver unit requires a complete design cycle from conceptual stage to prototyping. Translating the concept into design is possible with the knowledge of SMPS design rules and the set of topology specific equations. However, making sure the design will work as expected often requires rigorous testing and design changes on the prototype, especially in the SMPS designs. This is because, finding correct component values by theoretical calculation, for shaping the compensation

circuits, in general requires fine adjustment during hardware tests. Shaping the compensation is crucial for achieving an adequate phase margin at selected crossover points together with a high gain in DC. The surrounding noise, the noise generated by the SMPS itself and parasitic elements present in the hardware are the cause of it. In general, allowing phase margin (PM) above 70° and above 10 dB gain margin achieves a conservative and reliable design (Basso 2008). Exercising this is easier in simulation than with real hardware. Moreover, performance analysis with a varying battery terminal voltage (input perturbation analysis) and component values parameterisation to review the HT-Bank optimisation is easier in simulation than in hardware tests.

- Battery:

Energy drawn from batteries is not always equivalent to the energy consumed in device circuits (Rao, Vrudhula & Rakhmatov 2003) as some of the energy is dissipated or wasted as heat. Hence, understanding the discharge behaviour is crucial for optimisation. The state of charge of a battery can vary with the battery's age, discharge rate and the exposed temperature. The Li-Ion battery used in the design is an electro-chemical construct. Performance analysis of the battery using hardware was not possible due to the absence of hardware and lack of resources. A battery simulation model that can be electrically simulated for performance analysis is challenging, as it requires the coupling of electrical and chemical engineering effects to be included in the model. But equivalent circuit models have been published in the literature (Rao, Vrudhula & Rakhmatov 2003) that can be used with appropriate modifications in this research work. Simulating a suitable battery model at different stages of its capacity level and age with the system's discharge behaviour enables quantifying the number of complete functional cycles possible before the battery terminal voltage drops below an operational voltage threshold. The

discharge behaviour can be obtained from the combination of simulation results, measurement data and estimation.

The digital circuits and the data converters are assumed to consume the maximum power published by the manufacturers for the following reasons; their behaviour is always predictable and the simulation tool selected for performing this modelling task does not support digital or mixed simulations. Inclusion of their worst case power consumption in the power budgeting is adequate for the purpose of this research. Justification for the decision is documented in detail in Chapter 3; section 3.5.

2.10 Variant of application

System models are in general developed for simulating and analysing specific criteria of a component or system, meaning the component parameters that have no or a minimal effect on the interested simulation analysis are not included in the simulation model. This makes the developed model simple and discrete for fast simulation. These models are called equivalent circuit models. On the other hand, models developed following a topology specific methodology include almost all properties of the components and allow multi-functional simulations to be performed, meaning the same model can be used for power performance and functional performance. One such powerful application - reusability of models is demonstrated in this research. This work reused the models developed for power performance analysis to explore the possibility of enhancing the hardware to facilitate experiments in enhanced LRUT applications. Chapter 4 discusses this work, where medium frequency ultrasonic waves were employed for inspecting complex structures, which is impossible with a typical LRUT application.

2.10.1 Medium frequency application

Medium frequency ultrasonic waves in the high 100's of kHz were found to be adequate for inspecting complex structures such as critical aircraft components. Utilisation of high frequency in LRUT applications helps achieve higher resolution, which is required for complex structure inspection. Preliminary research work carried out (Haig & Stavrou 2012), using Finite Element Analysis (FEA), predicts the utilisation of an excitation frequency in the range of 150 kHz to 1 MHz to achieve sufficient resolution for inspecting the aircraft components of interest.

Hardware developed for a typical LRUT application cannot produce excitation signals of a frequency higher than 100 kHz without distorting the waveform. Moreover, the receive circuits within this hardware are not capable of handling the signals above approximately 150 kHz. The research approached this by first simulating the already developed subsystems of transmit, receive and power supply circuits with necessary modifications to suit the application. Having validated the modified designs for this enhanced application, the schematics were generated and hardware was prototyped.

The successful delivery of the hardware enabled researchers to successfully monitor the fatigue condition of aircraft components using a mid-frequency LRUT technique for the first time in TWI history. Extension of this work is underway, in which state of the art neural network techniques are being developed for aircraft components condition monitoring.

2.11 Summary

The concept of LRUT and the possibility of enhancing its instrumentation for portability, by means of system modelling were discussed. System modelling of this construct is challenging

as the system consists of multi-engineering discipline constructs and modelling the coupling between the engineering domains is difficult. Achieving acceptable accuracy of models is crucial for reliable analysis. Models used to optimise the power supply and battery for portability also have been utilised to extend the bandwidth of the device, enabling inspection of more complex components.

3 ENHANCEMENT OF PULSER-RECEIVER FOR LONG RANGE ULTRASONIC SYSTEM

3.1 Overview

Research work on the enhancement of the pulser-receiver for portability is documented in this Chapter. The Chapter firstly discuss how an appropriate load model was developed with acceptable accuracy. The Chapter then extends the discussion to the modelling and transient analysis of the pulser-receiver unit. The main components in the pulser-receiver unit, identified in Chapter 2 as components affected by the downsizing of the battery and the HT-Bank were modelled and simulated for power consumption analysis. Simulation data obtained were used in the decision making process of sizing the HT-Bank and the battery for optimum portability. An electrical domain simulation tool LTSpice was used for modelling and simulating multi-engineering discipline components that form the LRUT system.

3.2 Modelling of an Electro-Mechanical Load

In any system design and development process, knowing the system's load is crucial. In a LRUT application an array of piezoelectric transducers is the load, as discussed in the previous Chapter, and it is used as transmitters and receivers. These piezoelectric transducers are manufactured in a way so as to be used as shear transducers in LRUT applications. The mechanical motion/vibration under the influence of electrical excitation is illustrated in Figure 3-1. As indicated, an amplitude varying electrical excitation signal ($V_{EXCITATION}$) is applied across the transducer's electrical terminals, in the direction labelled as dimension – 1. It causes the piezoelectric transducer to deform diagonally in the direction labelled as dimension – 5, as indicated by the double-headed arrows. The term dimension 1 and 5 are used to denote the field-

directionality in piezoelectric ceramics terminology (BS EN 50324-1 2002). Physical parameters related to the 1-5 directions are used in the application and the modelling of these transducers.

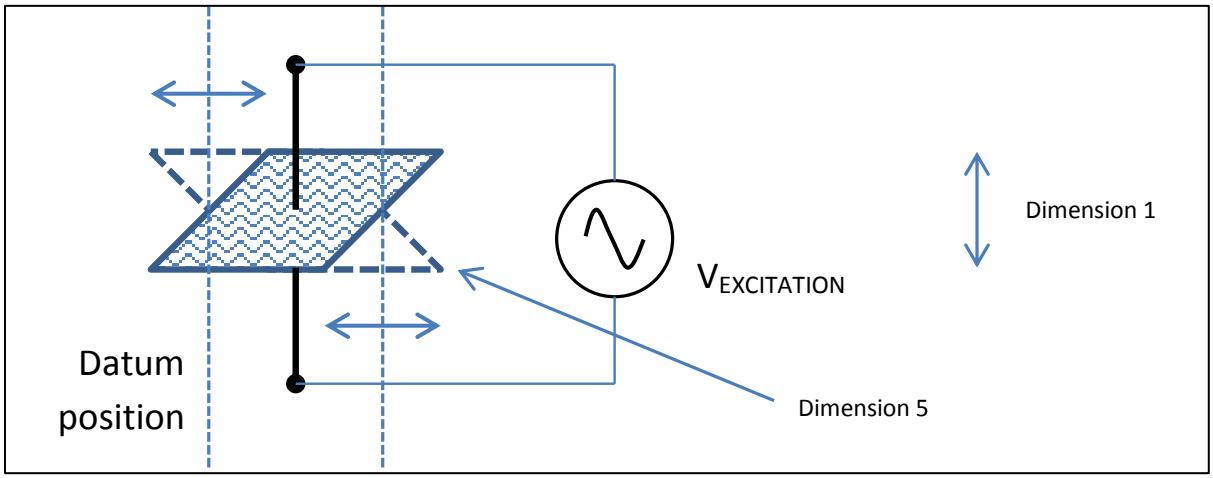
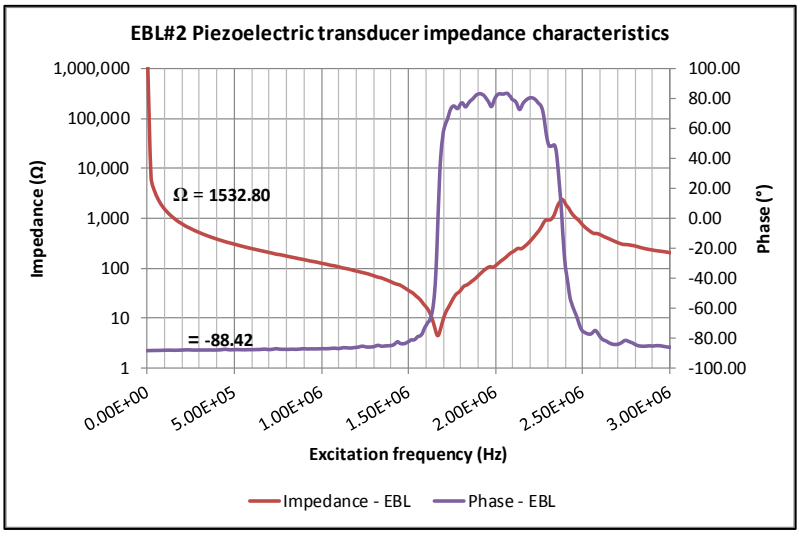


Figure 3-1 Shear mode (1-5) operation of a piezoelectric transducer

Piezoelectric transducers, of type EBL#2 are used in the LRUT application (EBL Piezoelectric precision 2010). These transducers, when used as active sensors (transmitters), have electrical impedance characteristics that influence the power consumption of their drive electronics – the transmit circuit in the pulser-receiver unit. Hence, the knowledge of this load impedance value is important for the power and functional analysis of the system concerned. In general, manufacturers of piezoelectric transducers provide their product's impedance characteristics in a graph format like the one shown in Figure 3-2 for an EBL#2 piezoelectric transducer manufactured by EBL. The table provided below the graph shows the test sample's specific physical characteristics, derived from measurements by the impedance analyser for inputted test settings/values.

Some important parameters required for modelling this transducer can be extracted from the graph and the generalised datasheet for EBL#2 (EBL Piezoelectric precision 2010). The extracted application specific parameters relevant for this work are presented in Table 3-1.

Further discussion on these parameters, together with some secondary parameters derived from these extracted parameters are included in the following section.



Sample			**Coupling Factor**	
Rectangular Plate/ Rod			f(min)(kHz)	: 1667.047
Electrode Distance {mm)	: 0.5000		f(max)(kHz)	: 2384.859
Electrode -Area (mm^2)	: 39.250		Q(mech)	: 38
			Q(3dB)	: 45.7
Measurement Conditions			B (3dB) (kHz)	: 36.499 (2%)
Averaging	: 1		keff (%)	: 71.51
Voltage (V)	: 0.5000		k15 (%)	: 74.94
Bias-Voltage (V)	: 0.000			
Capacitance Measurement				
Frequency (kHz)	: 1.000			
Capacitance (nF)	: 1.109808			
tand ($1e^{-4}$)	: 225			
eps (rel)	: 1597			

Figure 3-2 EBL#2 Piezoelectric transducer impedance characteristics and sample specific parameters (EBL Piezoelectric precision 2010)

Extracted parameter	Notation	Approximate value
Free capacitance	$C^{T}_{(at\ 1kHz)}$	1.1 nF
Resonant-frequency	f_p	1.7 MHz
Anti-resonant frequency	f_a	2.7 MHz
Impedance at 100 kHz	Ω_{100kHz}	1532 Ω
Phase below 100 kHz	$Phase_{100kHz}$	-89°
Coupling factor	k_{15}	75%

Table 3-1 EBL#2 piezoelectric transducer extracted parameters from manufacturer's data

3.2.1 Modelling of an EBL#2 piezoelectric transducer

A piezoelectric transducer is an electro-mechanical sensor and holds capacitive properties when excited below its resonant-frequency - f_p (BS EN 50324-1 2002). This is noticeable in the impedance curve presented in Figure 3-2, as the phase curve lay at approximately a negative 90° (~ -89°) phase, below f_p. The resonant frequency of a piezoelectric element is the excitation frequency where its impedance is the lowest. Anti-resonant frequency f_a is where the impedance peaks after the f_p.

There are number of equivalent circuit models published in the literature for piezoelectric transducers such as Butterworth-Van-Dyke (BVD), Masan, Redwood, Krimholtz-Leedom-Matthaei (KLM), Leach and Puttmer (Aouzale et al. 2007), (Jose, San & Antonio 2004), (Sherrit et al. 1999). These models provide powerful tools for the analysis and electrical simulation of piezoelectric transducer elements using modellers such as PSpice and LTSpice. With the exception of the BVD model, these models allow both the acoustic and the electrical part of a piezoelectric transducer element to be varied and analysed. Equivalent circuit models by Mason, Redwood and KLM are based on transformer topology. These models use a transformer primary to secondary ratio factor and reflected impedance characteristics for representing the coupling nature between the acoustic and electrical behaviour of the piezoelectric transducer. The Leach model is based on controlled source elements, and more simply represents the electro-

mechanical coupling of the transducer. Puttmer developed an equivalent circuit model based on the Leach model. He used a lossy transmission line model for the inclusion of losses in a low Q piezoelectric element in the model that the Leach model failed to demonstrate. This allows acoustic impedance mismatch losses to be analysed in broadband (low-Q) transducers. The Puttmer model also allows modelling of multilayer transducer arrangements with any number of piezoelectric and non-piezoelectric layers. This in general is normal in piezoelectric transducers assemblies for reducing acoustic impedance mismatches between the transducer and the test specimen. Piezoelectric transducers used in ultrasonic applications usually consist of a piezoelectric element and non-piezoelectric layers for encapsulation and acoustic impedance matching. LRUT-transducers include backing material (backing mass) and a face plate. It is crucial to include in the model the changes in characteristics of the piezoelectric caused by these extra fittings. Hence this research chose to use the Puttmer equivalent circuit model for modelling and simulating the LRUT-transducers.

The Puttmer model for the piezoelectric transducer of type EBL#2 requires a number of input parameters which are not usually published by the manufacturers. However they can be derived by a combination of measurements – using an impedance analyser and calculation using approximated equations 18, 20a, 20b, 36, 44, 50, 54 and 56 published in the British standards for piezoelectric transducers (BS EN 50324-1 2002). These equations are included in appendix A; Table A- 2 for reference. These approximate equations use shear mode specific parameters that can be extracted from the manufacturer's data sheet together with three experimentally obtained parameters; free capacitance (C^T), resonant frequency (f_p) and anti-resonant frequency (f_a), which were extracted from Table 3-1. The calculated model parameters are specific to an EBL#2 sheer-mode transducer of dimension 13 mm x 10 mm x 0.5 mm. These parameters were programmed

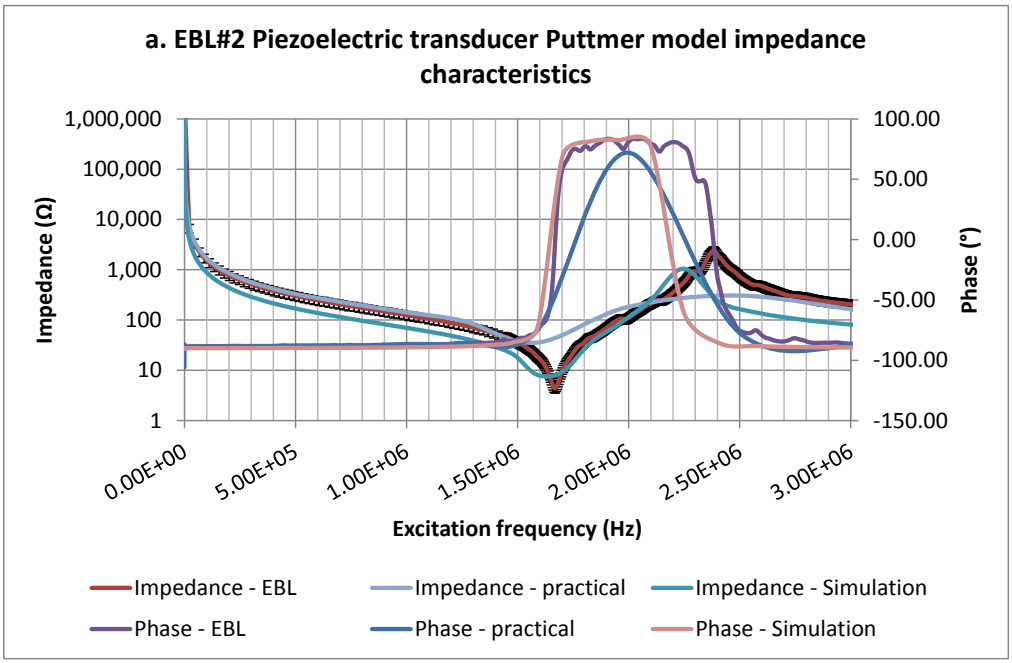
into the Puttmer model to form the simulation model for EBL#2 transducer. Both the parameter extraction method and the derived model are also presented in appendix A.

- Evaluation of the model

The developed model for an EBL#2 piezoelectric transducer of dimensions 13 mm (length) x 10 mm (width) x 0.5 mm (thickness) was simulated first and the simulation results were compared with the manufacturer's data; labelled as EBL and the experimental data; labelled as practical (obtained using impedance analyser) as presented in Figure 3-3; a. The practical results obtained matches EBL data below f_p, but deviates from the EBL data above f_p. The reason for this can be attributed to the cables and the test box used in the practical tests – which were not an exact match to the manufacturer's test setup and at higher frequencies the inductive nature of the cables used played a part in increasing the impedance. Differences in the impedance analyser test setup could have also influenced this, but require further investigation. Though the simulation results; labelled as simulation followed the trend of the EBL data, they did not closely match.

This developed simulation model was used as a baseline, and its parameters (refer to appendix A for details) were adjusted to achieve close mapping of simulation data within 20% of the manufacturer's data. The obtained results are presented (labelled as simulation) together with the manufacturer's data (labelled as EBL) in Figure 3-3; b with error bars on the manufacturer's impedance data set to ± 20% of the data value on each data point. This model was considered as a best fit Puttmer model for EBL#2 for this research. An accuracy within 20 % was considered acceptable as the manufacturer's datasheet publishes 20% tolerance (EBL Piezoelectric precision

2010). The trend of the phase graphs both for simulation and manufacturer's data closely match below the resonance frequency.



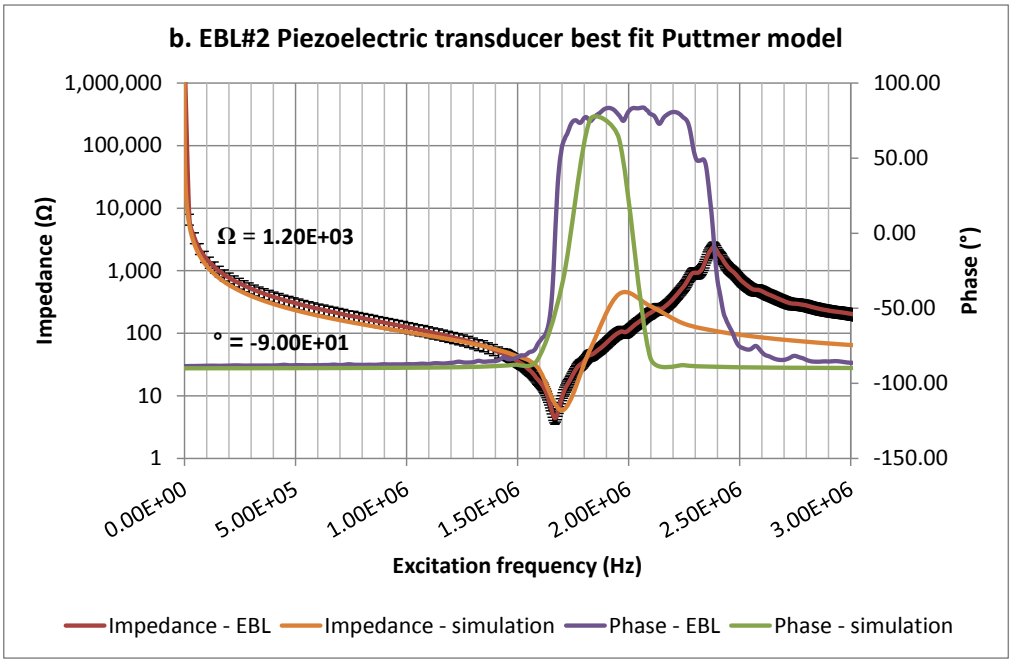


Figure 3-3 Equivalent circuit model evaluation for EBL#2 piezoelectric transducer

This simulation results are assumed to be accurate for further work. For the entire application range of 20 kHz and 100 kHz both the impedance and phase value stay within the 20% and 2% tolerance respectively. The reasons for the acceptance of this tolerance levels are further discussed and justified in the following subsection 3.2.2.

3.2.2 Modelling of LRUT transducer

Having developed an acceptable simulation model for the EBL#2 piezoelectric transducer, efforts had been made to develop a simulation model for the LRUT-transducer. It is a transducer assembly as shown in Chapter 2; Figure 2-1, built with a backing mass and a face plate for mechanical damping and protection respectively. Its schematic diagram is included in Appendix A; Figure A- 1. The backing mass is a stainless steel block, which was represented with a resister model in the simulation. Its value represents its acoustic impedance – $R_{SteelBackingMass}$ = 6 kΩ. This value was derived using the formula $R = \rho_{steel} \times A_z \times v_p$ (Leach 1994). Parameters ρ, A_z and v_p are the density of steel, area of the backing block and the phase velocity of sound in steel respectively (more information in section Appendix A 4). The face plate is alumina oxide (Al_2O_3) and it was modelled using a lossy-transmission line model. Clamping the transducer to the test specimen can also be simulated in the same way by connecting a resistor to the front face of the transducer. More details on this are given in appendix A. Thanks to the Puttmer model, all these extra fittings were cascaded together serially to form the LRUT-transducer model. The model parameters extracted for the faceplate are also included in appendix A; Table A- 5.

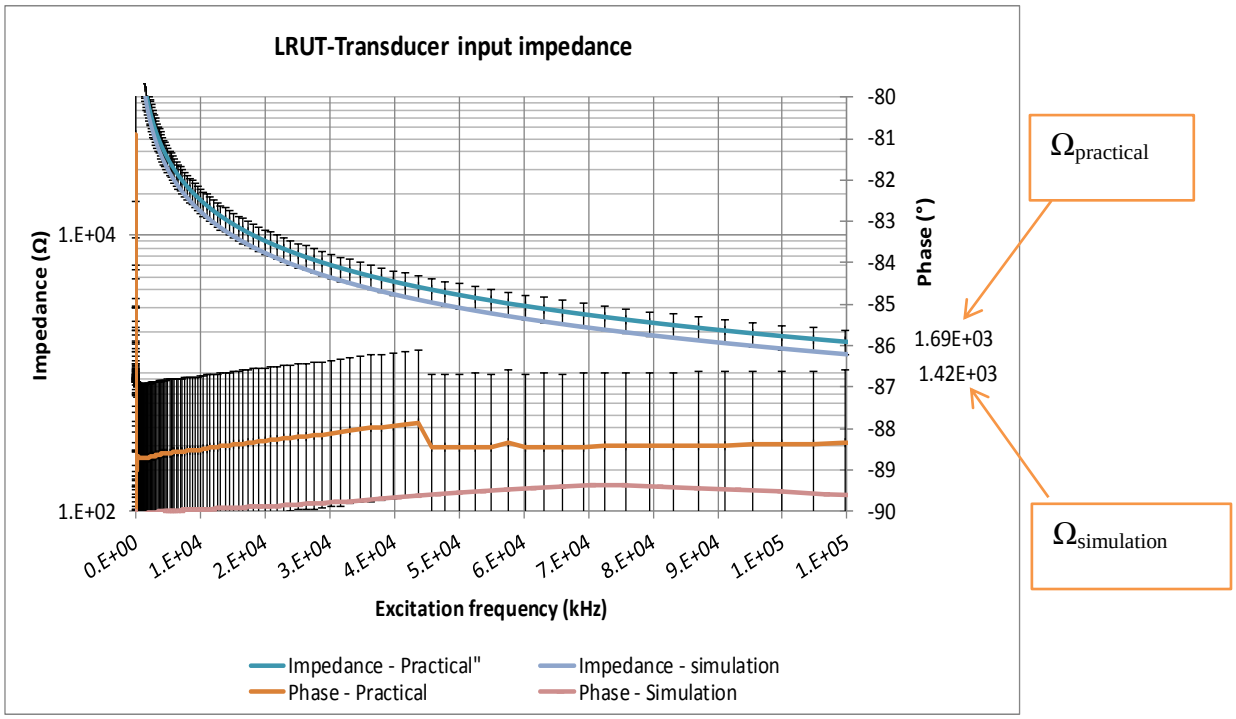


Figure 3-4 LRUT-Transducer impedance characteristics for the application range

Simulation results showing the impedance characteristics of the modelled LRUT-transducer are presented in Figure 3-4, together with the measurement data. The measurement data were obtained by characterising the LRUT transducer for the application range, using an impedance analyser. Error bars for impedance and phase graphs were set to ± 20% and 2% respectively on every measurement data point, indicating an accuracy within 20% on impedance was achieved. From the simulation results, it can be noticed that the impedance of an LRUT-transducer can be as low as 1140 Ω (1420 Ω ± 20%) at the maximum LRUT application frequency of 100 kHz. This loading effect in the capacitive term is approximately equivalent to 1.4 nF. The measurement results reveal the LRUT-transducer's input capacitance as 1.2 nF, showing less than 20% deviation between the simulation and measurement results.

- Acceptable accuracy of the model

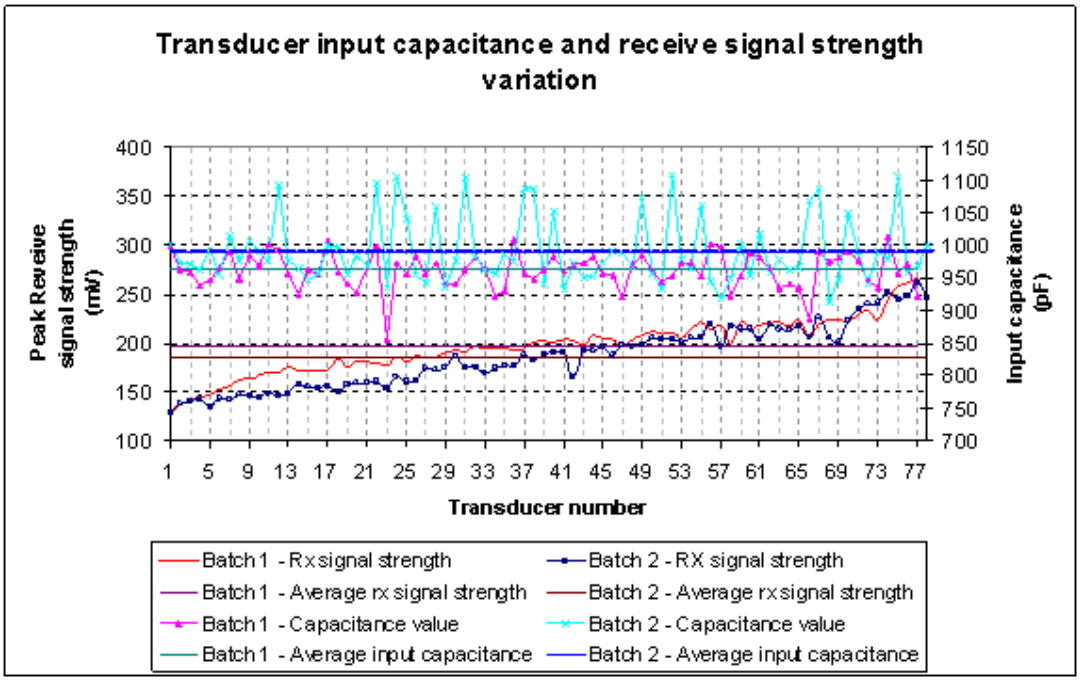


Figure 3-5 LRUT-Transducer production test results (Neal 2010)

Each LRUT-transducer, once produced undergoes a production test, in which its input capacitance and its transmitting ability are checked. Test results for two batches of seventy eight transducers are presented in Figure 3-5, showing above 20% variation in input capacitance (samples 68 and 75) and well over 40% deviation in signal strength. This implies that a tolerance of 20% is acceptable between the simulation and the practical results as even a larger deviation is possible in reality.

3.2.3 Transducer array

A transducer array of thirteen Teletest-transducers is the largest transducer-array size a single transmit channel is likely to drive in a typical LRUT application. Moreover, the operating setup requires the transducer array to be clamped or coupled to the test specimen. Coupling is achieved by the use of a bladder, wrapped around the transducer modules. When inflated the

bladder applies 60 psi (4 bar) pressure on the LRUT-transducers, providing necessary coupling/ clamping to transmit the ultrasonic energy into the pipe surface. Input impedance curves for clamped and unclamped transducer arrays obtained by measurement and simulation are presented in Figure 3-6. It shows applying pressure on the LRUT-transducers for clamping has not changed the input impedance value significantly.

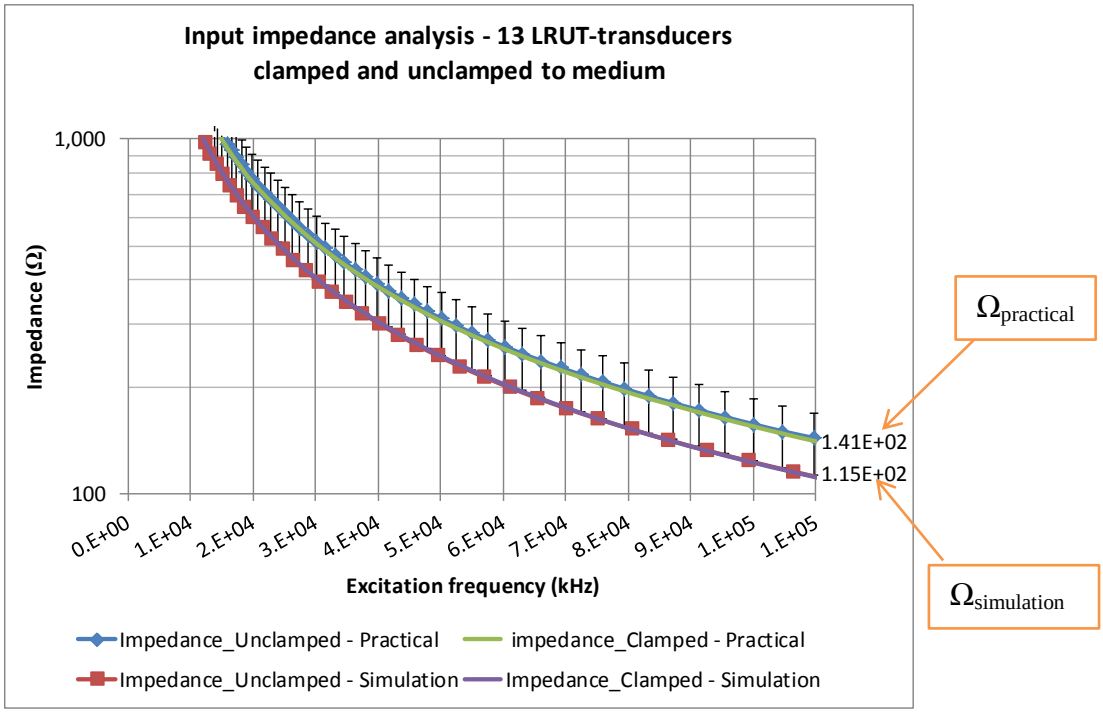


Figure 3-6 Impedance analysis of array size; 13 LRUT-transducers

The measurements showed an input impedance of the transducer-array is of a value of approximately 141 Ω at an excitation signal frequency 100 kHz. The simulation results showed a 20% discrepancy (115Ω). This is in good agreement, based on the 20% margin on tolerance. This part of the modelling work concluded that the minimum input impedance of a transducer array size of 13 transducers at 100 kHz is equivalent to 17 nF (with 20% tolerance taken into account 115 Ω ± 20%). The results presented covered the entire excitation frequency range

utilised in this LRUT application. The phase graph showed ~ 89° phase value (not included for clarity) with a discrepancy between the measurement and simulation within 2%.

3.3 Modelling of pulser-receiver unit

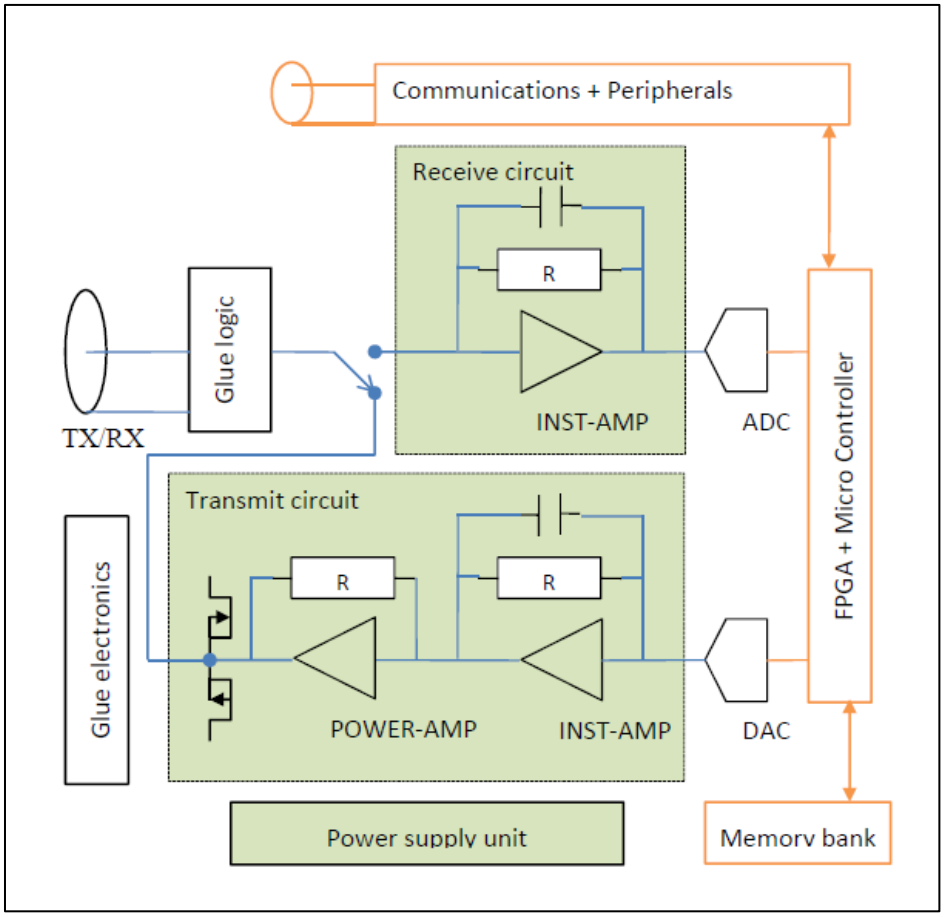


Figure 3-7 Simplified architecture of the pulser-receiver unit showing its main components

Having established knowledge on the loading effect caused by the transducer array, the drive circuits that excite the transducer array can now be discussed. This drive circuit is a subsystem in the pulser-receiver unit and is referred to as a transmit circuit. The pulser-receiver unit consists of a number of subsystems performing various functionalities that make the LRUT concept possible. The architecture of the pulser-receiver can be discussed in hardware terms with

the aid of Figure 3-7, which shows the main components of the pulser-receiver unit. The number of transmit and receive channels can be expanded to up to 24 channels in a typical system.

The pulser-receiver unit is the only assembly in the LRUT system that requires power to operate. Subsystem boundaries show the overall unit's operation in sub functionalities such as transmit, receive, power supplies, data converters, and digital logic. Data converters are the ADC (Analogue-to-Digital Converter) and DAC (Digital-to-Analogue Converter). The main components of the digital logic include the Field Programmable Gate Array (FPGA) and the memory bank. Boundaries also highlight the different types of electronics within each subsystem: analogue (green), digital (orange) and mixed-signal (ADCs and DACs). The following sections discuss the subsystems and the development of a model for them.

3.3.1 Transmit Circuit

The transmit circuit outputs an excitation signal that sufficiently excites the LRUT-transducers that consequently produce ultrasonic energy. Section 3.2 discusses the loading nature of the LRUT-transducer. The LRUT application uses the lower end of the ultrasonic spectrum in order to have a better mode control, low attenuation and couplant free tooling (Mudge 2006). This is well below the resonant-frequency of the LRUT-transducer. Hence this transducer holds capacitive properties towards their driving circuits - transmit circuit. The capacitive nature of these LRUT-transducers requires a high voltage stress to force them into oscillation (Jo da Silva et al. 2008), (Ramos, San Emeterio & Sans 2000), (Svilainis & Motiejunas 2006), (Xiaojin et al. 2007). The application requires a high voltage excitation signal at a high frequency (about a pulse of approximately 10μs duration at a frequency of 100 kHz) in order to achieve adequate resolution. A single transmit circuit was designed to support a maximum of 13 of these LRUT-

transducers, connected in parallel. The combined load impedance can be as low as 93 Ω, as discussed in section 3.2.3 (115Ω-20%).

Parameter	Symbol	Value
Supply voltage	V_S	±150V_{DC}
Excitation signal frequency	F_{EXT}	Min = 20 kHz; Max = 100 kHz
Excitation voltage	V_{EXT}	± 120 V (Peak ≅ 120 V)
Load	C_{LOAD}	Maximum of 13 LRUT-transducers ~ 17 nF
Input to transmit circuit	V_{IN}	0.1 V to 1 V Von-Hann windowed Sine wave (20 kHz to 100 kHz)
Voltage gain	G_{PA}	150 (43.5 dB)

Table 3-2 Transmit circuit specification

Parameter	Formula	Value
Slew Rate (SR)	$SR = 2 \times \pi \times F_{EXT} \times V_{EXT\_MAX}$	76V/µs
Total load Impedance (X_C)	$X_C = 1/(2 \times \pi \times F_{EXCIT_{MAX}} \times C_{LOAD})$	93.6 Ω
Peak load current (I_{PEAK})	$I_{PEAK} = SR \times C_{LOAD}$	1.3 A
Worst case power dissipation (P_{TX_WORST})	$P_{TX\_WORST} = 4 \times V_S^2/2 \times \pi \times X_C$	153 W

Table 3-3 Transmit circuit calculations

Previous experience and practical work carried out suggests that to sufficiently excite the LRUT-transducers to achieve the optimum inspection range, the excitation voltage should be as high as 240 V peak to peak ($240V_{pk\text{-}pk}$) with a frequency ranging from 20 kHz to 100 kHz. This is to generate sufficient acoustic energy to penetrate through the acoustic impedance mismatch between the transducers and the test specimen. Moreover having a high excitation signal increases the signal to noise ratio (SNR) (Ramos, San Emeterio & Sans 2000), (Svilainis & Motiejunas 2006). Due to the project constraints, the research focussed on the existing design, but developed a new layout to reduce noise and enhance functional performance. Hence, the modelling work was aimed at only analysing the power performance of the existing design of the transmit circuit. However, a design procedure was followed to understand the design and functionality constraints.

The specification of a transmit circuit is given in Table 3-2. Design specific calculations are included in Table 3-3 for component selection and evaluation (Cirrus logic 2009).

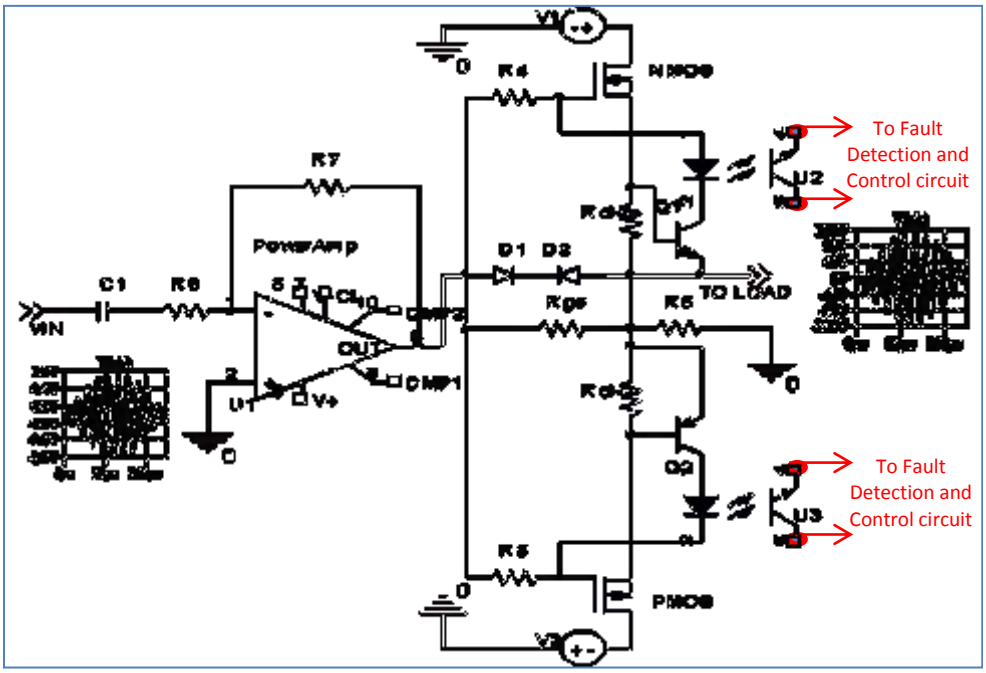


Figure 3-8 Simplified diagram of a transmit circuit

A simplified circuit diagram of the transmit circuit utilised is shown in Figure 3-8. It consists of an off-the-shelf power amplifier with a complementary pair of high voltage metal-oxide semiconductor field-effect transistors (MOSFET) of n-type and p-type at its output. Auxiliary circuits provide circuit protection should the I_{PEAK} exceed the protection level. The choice of power amplifier was influenced by its high voltage tolerance, high slew-rate and considerably low quiescent current consumption amongst the other commercially available power amplifiers (Cirrus logic 2009), (Cirrus Logic - Apex 2008), (Xiaojin et al. 2007). These power amplifiers however have a low output current capability – 350 mA pulsed. So the complementary power MOSFETs are included in the circuit to provide the high load current (I_{PEAK}) that is required, as shown in Table 3-3.

The functionality of the circuit is as follows: Resistor R_{GS} was chosen to guarantee the maximum required gate-source voltage - V_{GS} (obtained from MOSFETs datasheets) for the MOSFETs with the power amplifier output current limit. The protection circuit formed by U2-Q1-Rcl+ and U3-Q2-Rcl- provide the current limit protection should the load current exceed the maximum load current – 2.8A in the real hardware. Resistor R6 is a high value resistor, which provides additional protection for the power amplifier (limits the power amplifier output current) should the MOSFETs go open circuit. D1 and D2 are Zener diodes which limit the V_{GS} to the maximum specified V_{GS}. Power supplies denoted by V1 and V2 provide the required high voltage +150 V DC and -150 V DC respectively. Detailed operation of the circuit can be found in the literature (Steele & Eddlemon 1993).

- Modelling of Transmit Circuit

The transmit circuit is an analogue circuit and was modelled topology specific due to its important functional performance and this high power operation. Simulation work mainly concentrated on the transient behaviour analysis for analysing its functionality and power performance. Validation and evaluation performed on the individual component models were minimal or none as all the required models were obtained from the manufacturers and were assumed accurate. However, it was found that external components were required to make the simulation converge at high frequencies.

- Simulation of Transmit Circuit

Firstly, transient simulation of the transmit circuit was performed for varying load values, ranging from 1 nF to 42 nF. In this particular simulation a capacitor model provided in the LTSpice library was used to represent the load. This model greatly simplifies the

parameterisation of the load in simulation in contrast to the developed transducer-array model. The reason for using a higher capacitance value than the transducer load i.e. 42 nF, which is equivalent to 38 Ω when excited at 100 kHz, was to extend the analysis to the protection level that is set to 2.8 A load current in hardware. When a load value of capacitance 42 nF is excited with a 120 V; 100 kHz signal, in theory it would draw approximately 3A from the transmit circuit. The reason for setting the protection level to 2.8A, whereas the estimated current draw (I_{PEAK}) by the optimum load is only 1.3A is the short-circuit nature of the capacitive load at high *dv/dt*, which can generate a high current spike as it is initially charged. This current spike can be as high as 3 A ($I_{spike} = C_{LOAD}\frac{dv}{dt}$) and can cause a false over current trip if the protection level is set at a lower value. The input signal V_{IN} is a truncated sine wave voltage waveform with a centre frequency F_{EXT}. A raised-cosine function (Von-Hann) is used as a window function for truncating the sine wave with a pre-programmed number of cycles. One such waveform of envelop duration 200 μs is included in Figure 3-8, showing a 50 kHz, 10 cycle windowed waveform for V_{IN} as an example.

The transient simulation results, revealing power performance of the transmit circuit are presented in Figure 3-9; a, for varying load values. The data values highlighted are for a load capacitance value of 17 nF, showing the subsystem transmit circuit consumed approximately 200 W instantaneous power for sufficiently exciting the load without saturating the transmit circuit output. It also shows about 1.3 A peak current (I_{PEAK}) was drawn by the load as expected during this 120V; 100 kHz operation. Figure 3-9; b, reveals a significant amount of instantaneous power had been dissipated across the MOSFETs mainly due to resistive heating given by I^2R, where I and R are the load current delivered to the capacitive load and the R_{DS_ON} of the MOSFETs respectively. Power dissipated across the power amplifier was 3 W due to its quiescent power

(300 V x 10 mA) and the MOSFET gate current ~ 0.5 W. Power dissipation for driving 42 nF was as high as 500 W, which was not intended for typical applications.

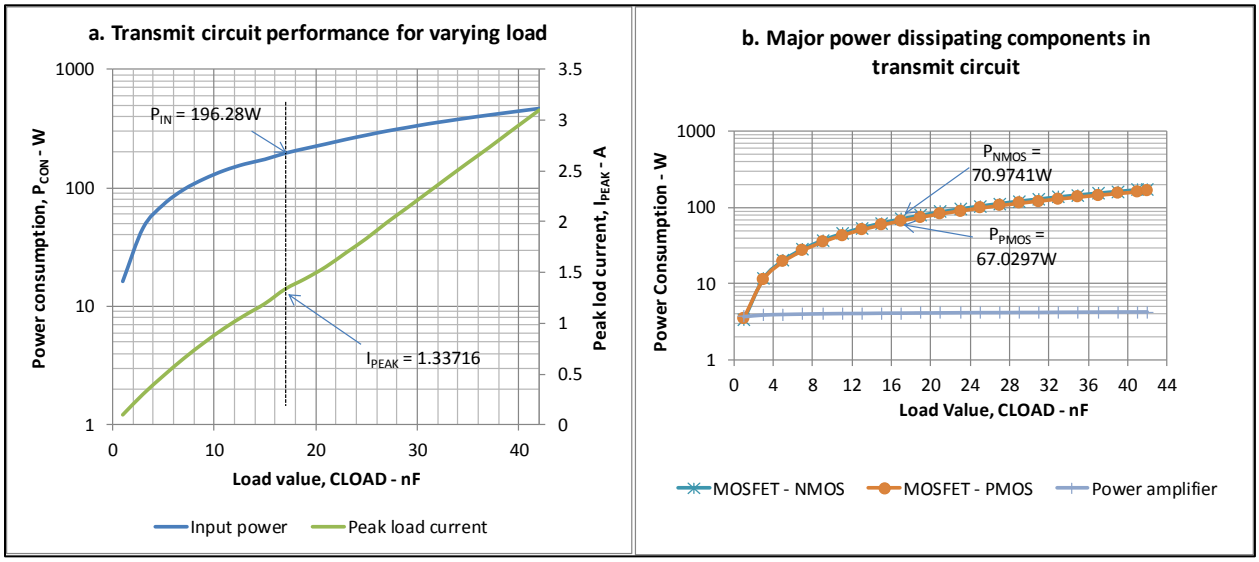


Figure 3-9 The graph shows all major power consuming components in the transmit circuit together with the power budget values

3.3.2 Power Budget Analysis with transducer array

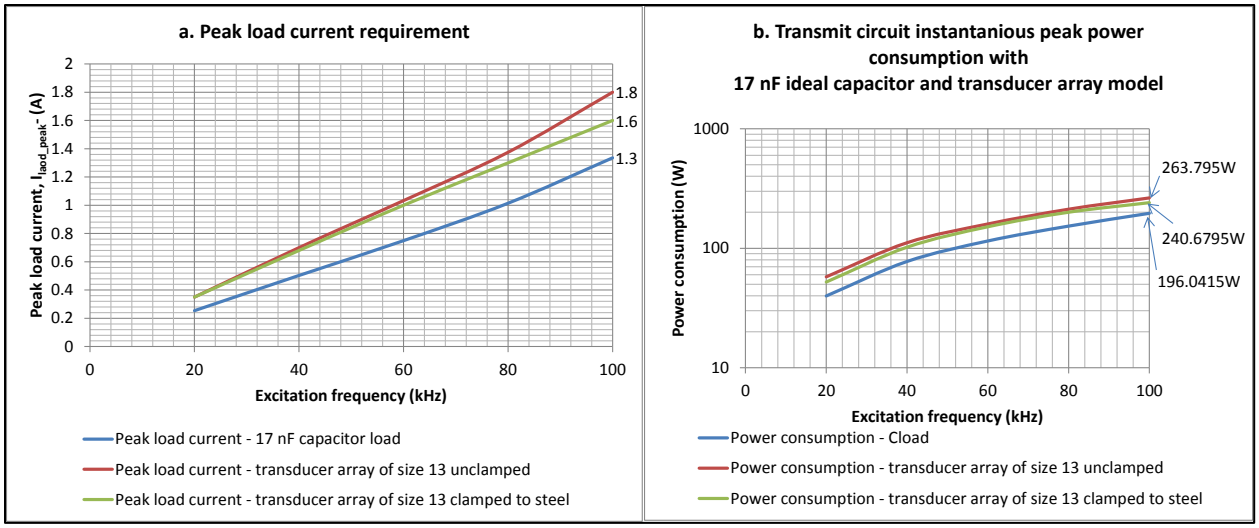


Figure 3-10 Transmit circuit power consumption revealed

The properties of the transducer array are not purely capacitive. Hence, the ideal capacitor used as the load in the transmit circuit analysis in section 3.3.1 was replaced with the load model developed for an array of 13 transducers and simulated for clamped and unclamped scenarios. The simulation results were then compared with those obtained for simulating the transmit circuit with an ideal capacitor of value 17 nF as presented in Figure 3-10; a, and b.

Results presented in Figure 3-10; a, show peak load current delivered to the ideal capacitive load of value 17 nF, unclamped and clamped transducer-array model, at a frequency from 20 kHz to 100 kHz. The load current drawn by the unclamped transducer-array was noticed to be as high as 1.8A, which is significantly higher than the load current drawn by the ideal capacitive load at an excitation frequency of 100 kHz. The instantaneous power consumption of the circuit (including power delivered to the load) was also higher (~ 265 W), as the power performance graph presented in Figure 3-10; b shows. The deviation in load current values for an ideal capacitor and the transducer-array is acceptable for the following reasons; an impedance value of a lossy transmission line, which was used to model a piezoelectric element, is given by $\sqrt{(j\omega L + R)/(j\omega c + G)}$, where the terms R. L and C are the characteristic capacitance, inductance and the pure resistance of the construct. Factor G is not supported by LTSpice and is set to zero. The AC analysis showed a phase of -88°, meaning it was not a pure capacitance. In contrast, the impedance of an ideal capacitor is given by $1/j\omega c$ and its phase is -90°. This shows that the assumption made that the transducer-array's input impedance is 17 nF (ideal), was only approximate and in reality its equivalent impedance is lower than an ideal capacitor, making it draw more current than an ideal capacitor. The difference between the unclamped and clamped transducer-array can be discussed using an analogy used in transformers – reflected impedance. The Puttmer model and all other models aforementioned (apart from BVD) are based on the

analogy of a transformer topology where the electro-mechanical coupling nature of the piezoelectric transducers was represented with the turn ratio between the primary and the secondary turns. When the acoustic impedance is added to represent the clamping effect in the secondary side of the transformer it is reflected back to the primary side of the transformer, making the primary current low with increasing acoustic impedance.

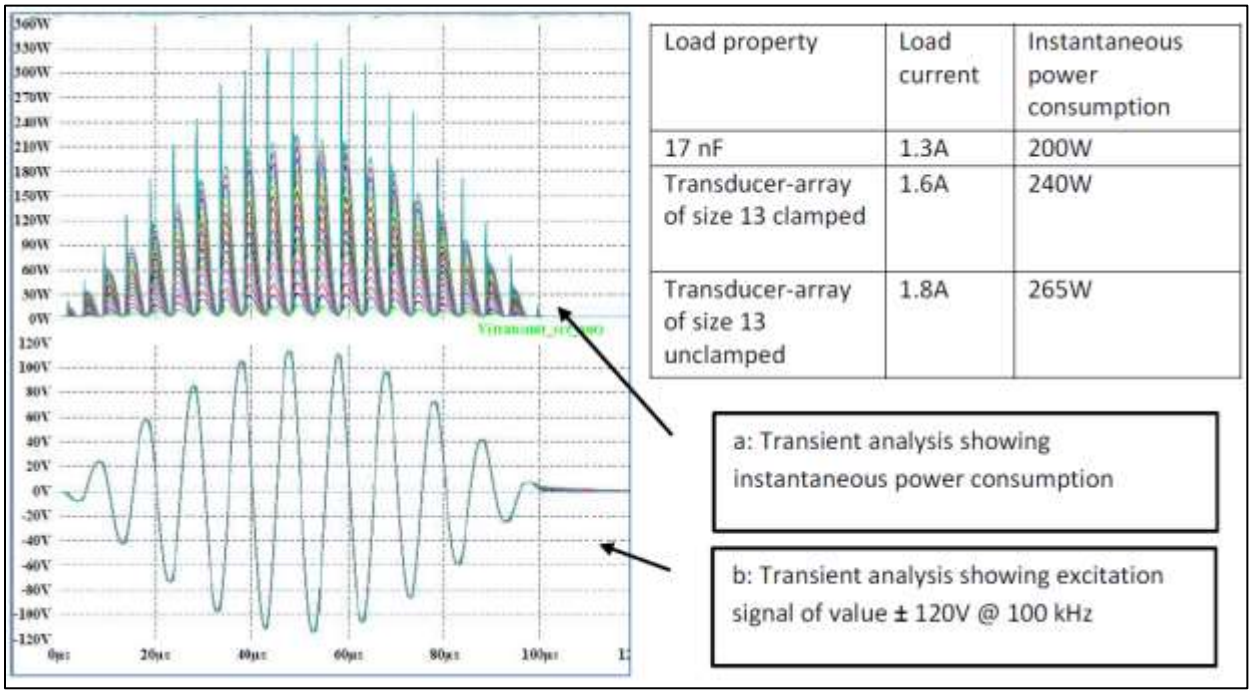


Table 3-4 Transmit circuit power consumption summary; ideal capacitor vs. load model

In summary, the instantaneous peak power consumption of the transmit circuit to sufficiently excite the transducer array of 13 LRUT-transducers at 100 kHz is about 265 W, compared to 200 W for a pure capacitive load of value 17 nF. This is shown in Table 3-4 with the instantaneous power consumption trace obtained by transient simulating the transmit circuit with 120V; 100 kHz excitation signal with a clamped load. The spike at the beginning of each power cycle is due to the short circuit nature of the capacitive load.

3.3.3 Receive Circuit

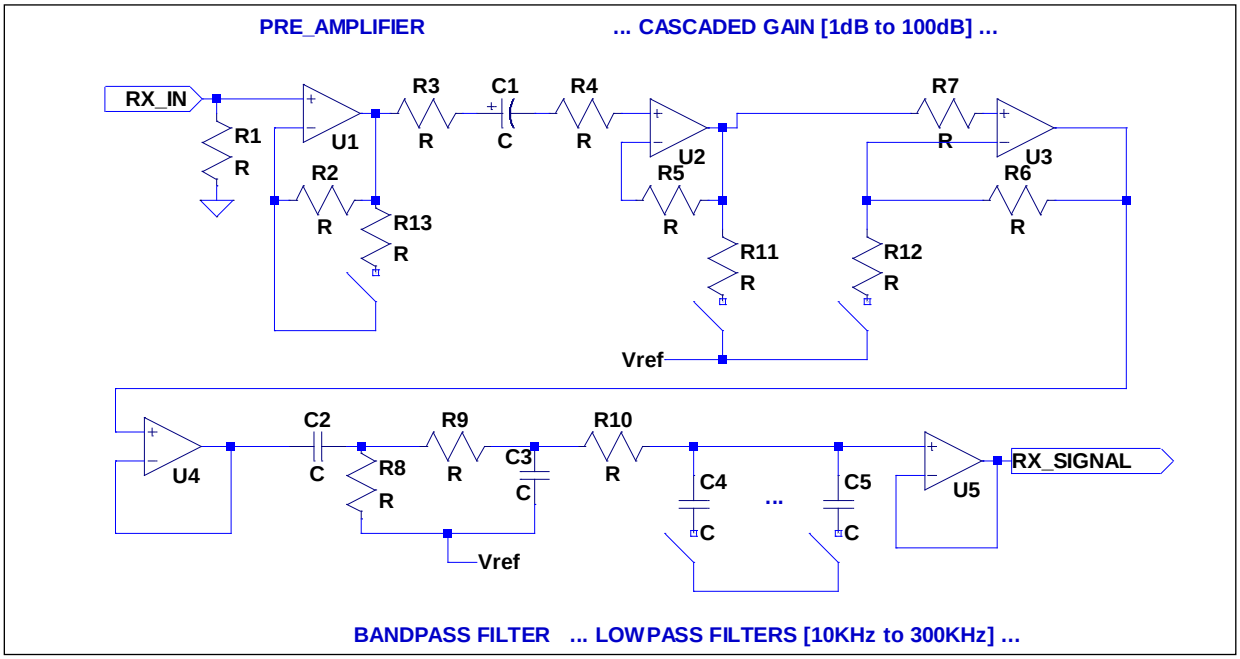


Figure 3-11 Receive circuit implemented in the pulser-receiver unit

The receiver circuit in the pulser-receiver was modelled to analyse its power performance in simulation, so that this sub circuit's power requirements can be included in the battery sizing exercise. As previously stated this circuit's performance does not affect nor is affected by the HT-Bank optimisation directly. This circuit design includes amplification and filtering stages for the received signals from the receiving transducers and outputs this signal to the digital signal processing subsystem. The circuit is a hybrid of operational amplifiers, analogue-switches and discrete components such as resistors and capacitors. The circuit has an overall gain of up to 100 dB that can be set in 1 dB steps from 20 dB. This is to make sure the received signal level is within the acquisition limit for the digital signal processing electronics. The selectable passive filters in the circuit enable adequate filtering. The circuit is powered with +/-5 V DC supply and it also utilises an auxiliary 2.5 V DC supply for level shifting (offset) in order to make the receive signal swing within 0V and 5 V. Figure 3-11 is a simplified schematic diagram, showing

the main components for one receive channel and a typical pulser-receiver unit is built-in with 24 of these circuits together with components/ circuits such as decoupling capacitors and localised power supply filter circuits.

- Modelling and simulation of Receive Circuit

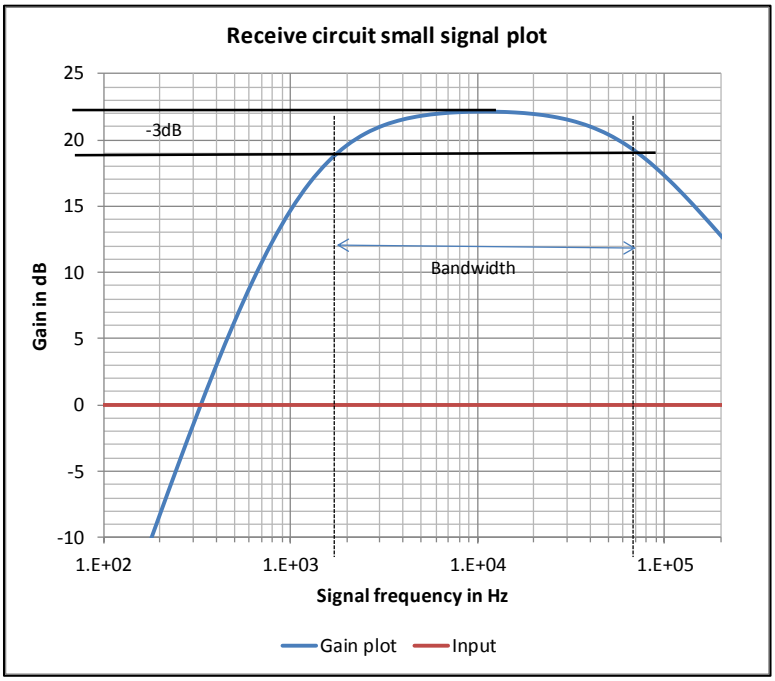


Figure 3-12 Bode plot showing functionality of receive circuit

The receive circuit is an analogue circuit and all its important components were modelled using topology specific models. The modelled receive channel was simulated for small signal AC analysis and transient analysis. The AC analysis facilitates the wide band response of the circuit to be plotted in a log scale – the Bode plot. AC analysis simulation results for the developed model are presented in Figure 3-12, with its functional configuration set to 20 dB overall gain and filter bandwidth 1 kHz for a high-pass filter and 75 kHz low-pass filter cut-offs. As can be seen, the gain achieved in the simulation was approximately 22 dB, instead of 20 dB, which has also been observed in the real hardware. A review of the circuit was recommended to

investigate the offset. The band-pass filter cut-off frequencies were noted to be 1.8 kHz and 70 kHz. The circuit is implemented with a passive filter circuit of a simple single stage R-C configuration; hence sharp roll-off was not expected. This functional performance analysis enabled the evaluation of the model developed.

- Power Budget Analysis

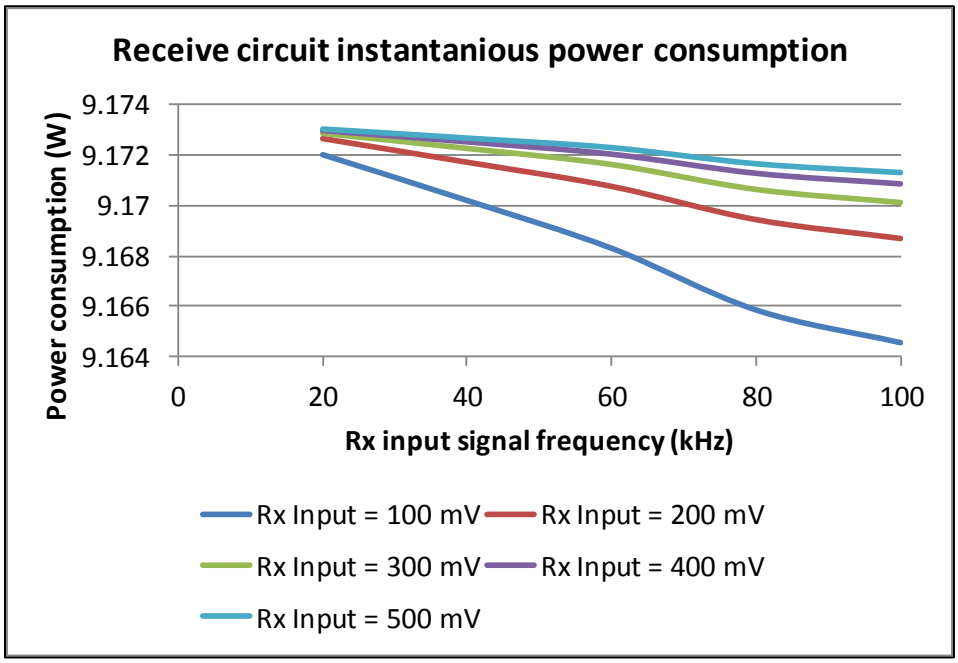


Figure 3-13 Receive circuit power performance

Power performance analysis of the receive circuit was performed using transient analysis. The simulation was carried out with the circuit's input signal amplitude being parameterised from ±100 mV to ±500 mV in ±100 mV steps (primary parameterisation) while its frequency was parameterised from 20 kHz to 100 kHz at each primary parameterisation step. In all the simulation steps the overall circuit gain was set to 30 dB and the filter settings were set to 1 kHz high-pass and 75 kHz low-pass cut-offs. This allowed the circuits to be analysed for out of range signals as well as distortion scenarios. Power consumption results obtained from the transient

analysis are presented in Figure 3-13 showing the circuit consumes around 10W power for all varying input signal levels across the application specific frequency range. However, there is a noticeable change in power consumption when the input signal strength is ±500 mV, compared to the power consumption for an input signal strength ±100 mV. This is because for a higher input signal voltage amplitude of 500 mV, the receive circuit saturated as a consequence of the high voltage gain setting of 30 dB set during this particular transient simulation.

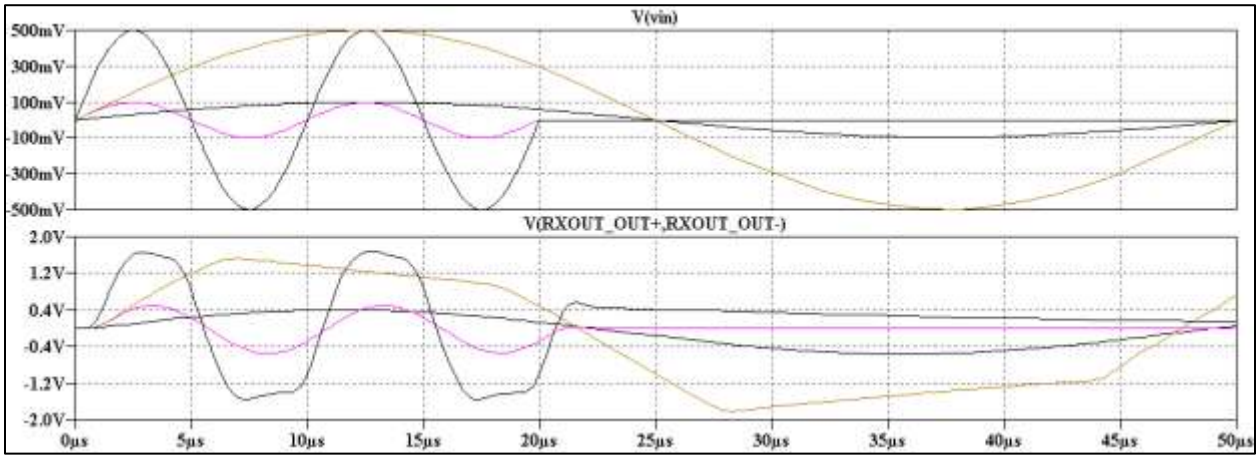


Figure 3-14 Transient simulation depicting the clipping of the receive circuit output

The circuit's output amplitude level, labelled as (V(RXOUT_OUT+,RXOUT_OUT-)), for input signal amplitude levels ±100 mV and ±500 mV, labelled as V(Vin), for the input signal frequencies 20 kHz and 100 kHz, are presented in Figure 3-14 for justifying the reasoning. It shows severe clipping of the output signal for an input signal amplitude of value ±500 mV. Bringing the amplifier gain down to 20 dB helped to prevent clipping. Power consumption is slightly higher at lower frequencies due to the longer signal period.

The simulation results concludes that the change in power consumption is insignificant throughout the spectrum and budgeting for 10 W per receive channel is adequate for battery

performance analysis. Optimisation of these circuits is recommended to achieve better power performance.

3.3.4 Power estimation of Digital logic and Data Converters

The power consumption of the digital logic has been assessed in two states; static when there is no signal activity in the circuit and dynamic which occurs with a signal transition. The static power consumption is in general invariant and it depends on the supply voltage and the quiescent current drawn by the architecture of the integrated circuits and the other electronics. Manufacturer's datasheets normally provides this value for their devices. However, the dynamic power consumption can vary according to functional specific signal transitions and glitches. An analytical model for approximately calculating the dynamic power consumption, $P_{dynamic}$ of digital integrated circuits, was developed by (Quang, Deming & Martin 2010), which expresses $P_{dynamic}$ by the equation $P_{dynamic} = \frac{1}{2} f_{clk} V_{dd}^2 \sum_{i=1}^{N} C_i S_i$, where f_{clk} and V_{dd} are the clock frequency and the supply voltage. Parameter C_i is the load capacitance for gate í and S_i is the switching activity for gate í. Manufacturers do not normally publish parameter C_i and working out S_i is difficult in large circuits. Work published on equivalent circuit models for power estimating ADCs by (Lauwers & Gielen 1999) suggests that values for C_i can be replaced by the channel length the technology of the integrated circuit was fabricated with; e.g. semiconductor chip manufacturer Texas Instruments used 0.1 micrometre technology for fabricating the ADCs of manufacturer's part number ADS803 (ADS803 2002), utilised in the pulser-receiver circuits).

The major power consuming digital circuits in the pulser-receiver circuit are the FPGA and the memory bank. There are ways to estimate the total power consumption of the FPGA for a specific design such as using a vendor specific power estimator (Xilinx power estimator 2010), X-Power-Analyser (power estimation tool integrated in the Electronic Design Automation tool

provided by the FPGA manufacturer) and by doing a measurement. Using (Xilinx power estimator 2010) is a quick method, which allows an approximate/ typical estimation of power consumption of the FPGA and does not require a netlist of the design. X-Power-Analyser predicts FPGA power consumption more accurately, but requires the design netlist to be inputted. Measurements require hardware for an evaluation. This research estimated the static (quiescent) power from the manufacturer's data sheet (Xilinx Spartan-3A DSP 2011) values as presented in Table 3-5. The dynamic power was estimated using (Xilinx power estimator 2010).

Description	Supply level	Quiescent current	Approx. quiescent power
FPGA Internal supply	1.2 V	900 mA	1.2 W
FPGA Auxiliary supply	3.6 V	145 mA	0.5 W
FPGA Output driver supply	3.6 V	5 mA	0.02 W
Total			2 W

Table 3-5 FPGA static power estimation

For dynamic power consumption, it was assumed 100% of resources in the FPGA were utilised and the logic and I/O ports were toggled 100% at a switching frequency of 125 MHz. Dynamic power dissipation values produced by (Xilinx power estimator 2010) are presented in Figure 3-15, showing approximately 4W power can be consumed due to worst case signal transitions, totalling 6W power consumption for the FPGA only. The bottom left hand corner of Figure 3-15 shows that the I/O ports and the internal logic accounts for 41% and 53% of the total dynamic power consumption. Transceivers were not used in the application.

Memory devices used in the application were supplied with 3.3 V and their absolute maximum power dissipation was rated at 1 W, from the manufacturers data sheet (Integrated Silicon Solution Inc 2012). The memory bank is likely to be employed with 6 of these devices for adequately storing the data, making total power consumption of the memory bank 6W.

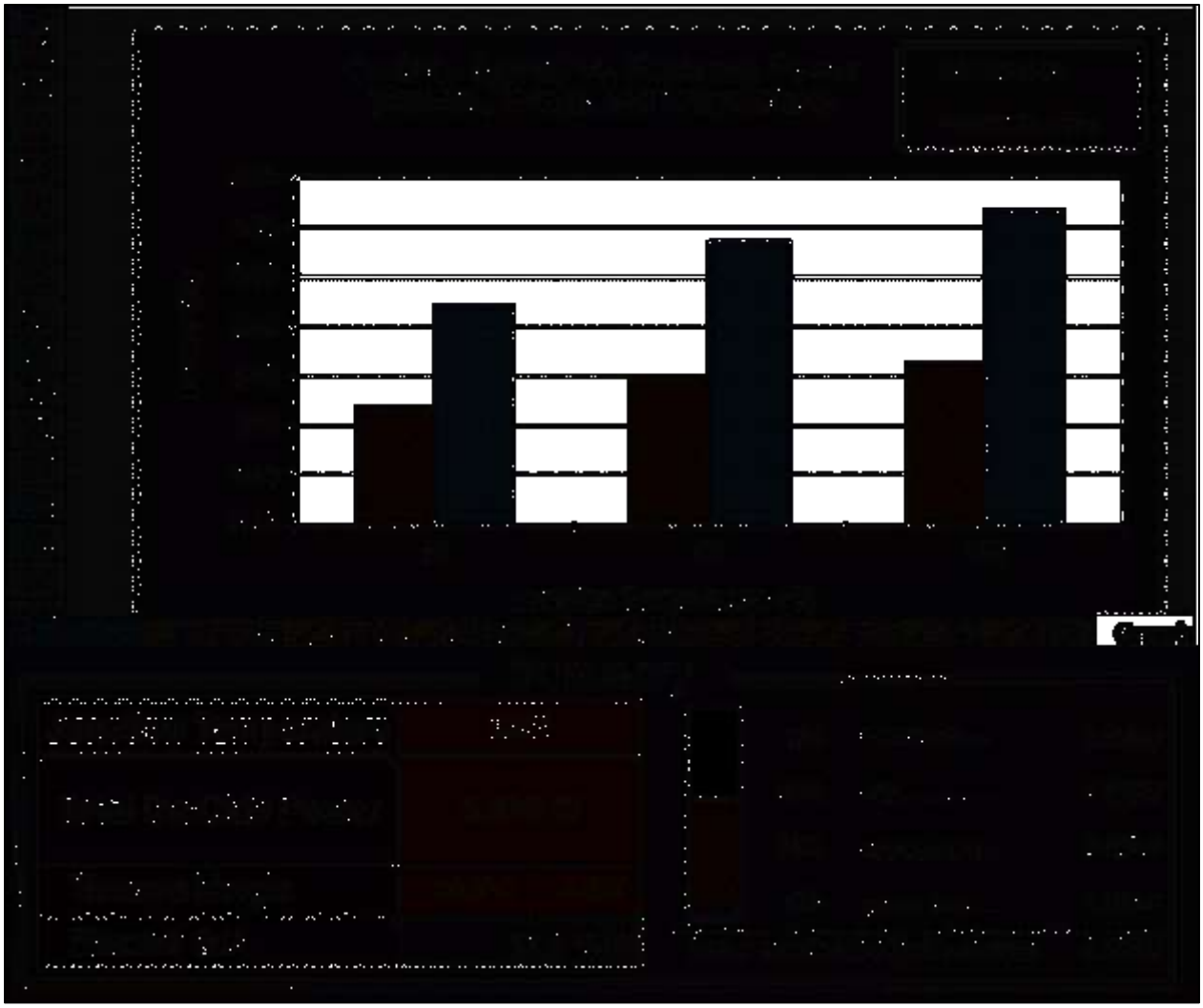


Figure 3-15 FPGA dynamic power consumption

Data converters are mixed signal components, having an analogue front-end and a digital back-end for ADCs and a digital front-end and an analogue back-end for DACs. The chosen simulation tool LTSpice neither supports digital simulation nor mixed-signal simulation, requiring an analogue equivalent circuit model for data converters for circuit simulation. The goal of optimising the portability of the pulser-receiver unit does not require a functional simulation of the data converters. Hence the research only concentrated on analytically

predicting the power consumption of the ADCs and DACs, so that the battery capacity analysis can include this data. The HT-Bank optimisation does not compromise this circuit operation.

Topology specific models for data converters ADCs and DACs in general do not exist, due to the complex nature of the mixed signal circuits. However, efforts had been made by researchers to develop equivalent circuit models for ADCs (Lauwers & Gielen 2002), (Lauwers & Gielen 1999) and DACs (Brodersen & Al-Ali 1989), (Thomas, Linda & Arthur 1982) for power consumption analysis. An analytical model published by (Lauwers & Gielen 1999) was found to be suitable for this research due to its top level approach in assigning model input values that can easily be found in ADC manufacturer's datasheets. This model expresses the power consumption P_{ADC} using the formula $P_{ADC} = \frac{V_{dd}^2 L_{min}(F_{sample}+F_{signal})}{10^{(-0.1525\times ENOB+4.838)}}$, where V_{dd}, L_{min}, F_{sample}, F_{signal}, ENOB are the supply voltage, channel length (fabrication technology), sampling frequency, signal frequency and effective-number-of-bits of the ADC respectively. All input parameters that are required to solve this equation for estimating the power consumption of the ADC used in the design were extracted from manufacturer's datasheet (ADS803 2002). Based on the extracted values V_{dd} = 5 V, L_{min} = 1.6 μm, F_{sample} = 5 MHz, F_{signal} = 50 kHz, ENOB = 11 the power consumption P_{ADC} of the ADC used is 139.6 mW. The datasheet claims that this particular device consumes 115mW.

Macro models for DACs on the other hand that can be simulated in a Spice environment have been published based on current switched R-2R ladder configuration by (Thomas, Linda & Arthur 1982) and later by (Brodersen & Al-Ali 1989) who simplified the model by using an ABM current source and switch models. These models allow transient simulation, but without knowing the architecture of the specific DAC it is difficult to model and simulate the developed

model for power analysis. This research used the DAC datasheet power consumption values for the calculation.

Component	Power consumption	Number of devices	Total power consumption	Operational time
ADC	140mW	24	~ 3W at 3.3V	120 s
DAC	120mW	24	~ 3W at 3.3V	1.1ms

Table 3-6 Power budgeting of data converters

The pulser-receiver unit in a typical application would consist of 24 of these ADCs and DACs, giving a total power consumption of 3W for ADCs and DACs. The ADCs are assumed to be turned on during the whole LRUT cycle, and the DACs are only turned on briefly for 1.1 ms per excitation as summarised in Table 3-6.

3.4 Power Supplies

One of the tasks of the research was to design and prototype an appropriate power supply subsystem for providing the required voltage levels and sufficient current to all the subsystems that require power. The pulser-receiver unit required DC power levels of +5 V, -5 V, +24 V, -24 V, 12 V and ±150 V. All these power domains were modelled, simulated and the simulation circuits were translated into design schematics for prototyping. This work is documented in (Parthipan 2010) and one of these power domains, ±150 V provided by HTPSU, is repeated in this thesis due to its importance in the pulser-receiver performance and portability. This work has also been published in a journal paper (Parthipan et al. 2011). The pulser-receiver is also equipped with other auxiliary power supplies; 3.3 V, 1.8 V and 1.2 V, for providing power to digital electronics. These power domains were provided utilising off-the-shelf SMPS power modules from Texas Instruments (PTV05010W 2005), hence these have not been modelled or

evaluated. The choice of these power modules were constrained by the reference design that the digital electronics was based upon.

3.4.1 High Voltage Power Supply

This high voltage power supply – HTPSU, provides steady +150 V DC and -150 V DC supplies to all 24 transmit channels in a typical pulser-receiver unit. Simulations carried out on the transmit circuits with their load (see section 3.3.1 - Table 3-4), revealed each transmit channel on average consumes approximately 0.265 Joules of energy per excitation of a duration of 1 ms, based on the simulation data of an instantaneous peak power of 265 W (265 Js^{-1} x 1 ms = 0.265 J) for its heaviest load condition; 13 of LRUT-transducers excited at ± 120 V; 100 kHz. This totals to approximately 6.5 W average power (0.265 J x 24 channels/1 s), consumed by all 24 transmit channels, when each one of them excites the heaviest load. The loading on the HTPSU caused by the transmit circuit activity is a pulsed action, as shown in Figure 3-16. Each pulse-duration (T_{PULSE}) can be as long as 1 ms and the PRR (T_{PRR}) in a typical application is 10 Hz (0.1 s). A typical LRUT application utilises a Von-Hann windowed sine wave as shown in Figure 3-16 for the excitation of the transducers, meaning the stated peak power consumption is only reached at the main lobe of each envelop. However this power analysis assumes the pulse envelope is a tone-burst operation, whereby the current drawn from the +Ve and –Ve output rails of the source – HTPSU, is 1.8A on every single cycle within an envelope. This is symbolically shown in Figure 3-16, using red and green dashed-lines.

For the generation of the short high-power pulses like this, while maintaining the supply voltages to the transmit circuit at ±150V, power converters based on capacitor discharge are used, with appropriate recharging circuits to maintain regulation.

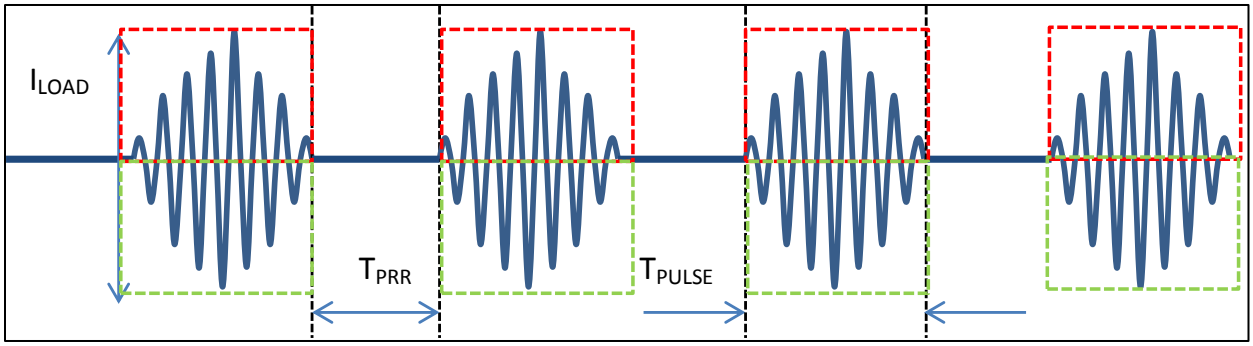


Figure 3-16 Pulse load application of pulse-receiver unit

- Allowable ripple

Ripple on the HTPSU output rails +150 V DC and -150 V DC, can jeopardise the quality of the excitation signal and consequently degrade the ultrasonic performance of the LRUT system. It can also cause oscillations in the transmit circuits. Hence providing a steady supply is crucial. With reference to the transmit circuit schematic depicted in Figure 3-8, the complementary circuit arrangement of the MOSFETs is a current source whose transient behaviour relies on its drive signal V_{GS}, the output of the power amplifier. The PSRR of the power amplifier used in the application is about -110 dB at 10 Hz. A ripple at 10 Hz is expected in the supply rails due to the pulse repetition rate (PRR) used in the application for averaging purposes. Allowing 15 mV ripple (0.01%) on the V_{GS} ($V_{GS\text{-}RIPPLE}$) without compromising the ultrasonic performance, requires careful design of the output stages of the power supply, ensuring the voltage drop on the HTPSU output rails caused by the excitation of the LRUT-transducer array is kept below the acceptable ripple voltage $V_{RIPPLE\text{-}HTPSU}$ level.

The acceptable ripple voltage $V_{RIPPLE\text{-}HTPSU}$, allowed on the HTPSU output rails can be calculated using the Equation 3-1, where G_{PA} is the gain implemented in the transmit circuit

amplification stage. Substituting the values $V_{GS\text{-}RIPPLE}$ = 15 mV, G_{PA} = 150 and PSRR = -110 dB in the equation, provides the figure for $V_{RIPPLE\text{-}HTPSU}$ of around 30V.

$$V_{RIPPLE-HTPSU} = \frac{V_{GS-RIPPLE}}{10^{PSRR/20} \times G_{PA}} \qquad \textbf{Equation 3-1}$$

- Topology selection

Normally, power to this type of pulsed load is provided using bulk capacitor-bank storages. With this configuration, energy is stored in a bulk-capacitor bank and discharged to the load when demanded. Topping up of the capacitor-bank is required to maintain the regulated voltage level at its terminals between repetitive load pulsing. Push-Pull converter topology and flyback topology are commonly used in rapid capacitor charging processes. A Push-Pull topology is generally used in applications where the power requirement is higher than 200 W. A single stage flyback topology was selected for this application for its simplicity, size, low cost and its widespread use in power applications that require power levels less than 200 W. Note that the power rating of the HTPSU is of the order of 10W for this particular application.

- Bulk capacitor bank HT-Bank size

Having worked out the values for $V_{RIPPLE\text{-}HTPSU}$, and the peak load current $I_{load\text{-}peak}$ demanded from the +Ve and the –Ve rails separately by each transmit channel, the bulk-capacitor value that can be attached to the individual output rails can be calculated using the capacitor discharge equation It = CV. Here the parameters I and V are the $I_{load\text{-}peak}$, and the $V_{RIPPLE\text{-}HTPSU}$ respectively. Parameter C is the minimum value of capacitance required on the supply rails individually per transmit channel to store sufficient energy to prevent the supply rail

voltages from dropping below ± 120 V. Substituting values of 1.8 A, 1 ms and 30 V for I, t and V, the capacitor value C can be calculated as 60 μF per rail to serve a single transmit channel.

The real hardware is required to support 24 transmit channels, hence the capacitor bank was equipped with 1600 μF capacitor bank per rail, totalling 3200 μF for +Ve and –Ve supply rails as a conservative measure. This is significantly higher than the assumed requirement of capacitor bank size of 1000 μF discussed in Chapter 2; section 2.5. So the possibility of reducing the size of the capacitor bank, to less than 3200 μF for enhancing portability was not recommended as an option.

- Power supply design

The design process of this power supply employed modelling of the design and simulating it for validating its performance prior to prototyping. The design specification of the power supply is summarised in Table 3-7 and the schematic, used for the simulation is shown in Figure 3-17.

Parameter	Symbol	Value
Input	V_{TRANS}	Max 16.8V; Typical 15V
Power supply Output	V_{OUT}	± 150 V_{DC}
Maximum Power output	P_{OUT_MAX}	13W +50% ~ 20W
Capacitor bank	C_{BANK+}, C_{BANK-}	1600 μF per rail

Table 3-7 High voltage power supply specification

The operation of the flyback power supply is explained in (Christophe 2008). The only difference between the work reported in (Christophe 2008) and this work is the split power-supply design. This was achieved using a centre-tap transformer (Parthipan T C8117 2010). With reference to the schematic diagram shown in Figure 3-17, its operation can be described as follows: in flyback topology, the transformer T1 is used for maximum energy storage purposes.

Hence, it is built with air gaps in the core to ensure maximum flux changes. At turn on of the power switch Q1 the primary current (I_{pri}) in the primary-inductance (L_{pri}) ramps up, with the consequent magnetic flux storing energy in the transformer core. A voltage is induced across the secondary winding (L_{s1}, L_{s2}) of a polarity such that D1 and D2 are reverse-biased. Hence, no current flows in the secondary circuit. When Q1 is turned off the primary current I_{pri} drops to zero and the voltage across secondary windings L_{s1} and L_{s2} reverse, allowing D1 and D2 to conduct current that in turn charges capacitor banks C_{bank+} and C_{bank-}.

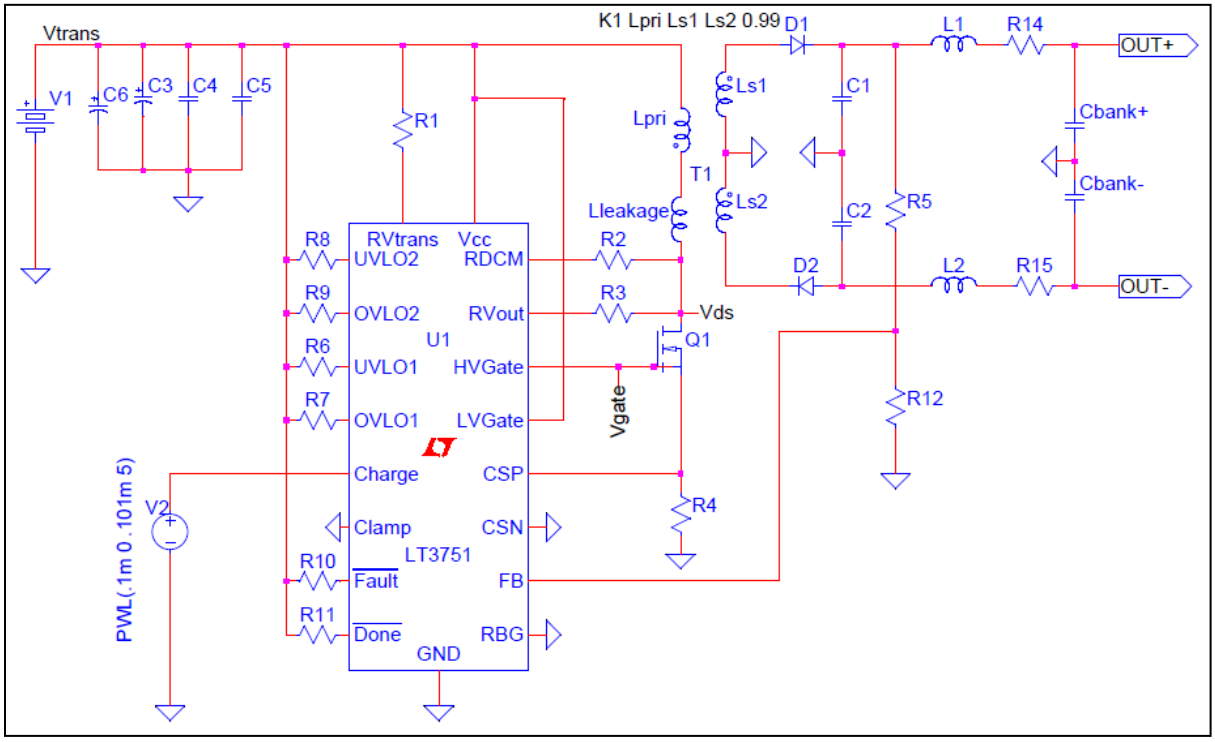


Figure 3-17 Simulation model of CCPS

The circuit was implemented with the aid of a commercial capacitor charging controller IC (LT3751 2008). The designed circuit stepped through three operating modes: namely charge mode, regulation mode and discharge mode in the process of charging and regulating the capacitor bank HT-Bank. In the charge mode the HT-Bank (C_{bank+} and C_{bank-}) is charged from

zero volts (if it is empty) to the set output voltage ±150V DC. In this mode the circuit draws a high primary current (I_{pri}) from the primary battery source. The power consumption is high during the charge cycle. In the regulation mode the output voltages are maintained with very small primary currents drawn to compensate for leakages in the capacitor bank. The third mode is a discharge mode where the load (transmit circuit) draws current from the HT-Bank. No current is drawn from the primary source at this stage. However as the HTPSU is in regulation mode the discharged energy from the HT-Bank requires topping up (regulator mode) before the next excitation cycle (PRR is 10 Hz in a typical application) without compromising the PSRR aforementioned. This time the current will be drawn from the primary source. The operation is documented in more detail in (Parthipan 2010) and the datasheet of the controller IC (LT3751 2008).

- Operating parameters and components selection

Design Parameters	Formula used in the calculation	Calculated Value	Selected Value
Transformer ratio N	$N \leq V_{OUT}/V_{TRANS}$	10	10
Peak primary current I_{PRI}	$I_{PRI} = \frac{P_{OUT(AVG)}}{Efficiency} \times \left(\frac{1}{V_{TRANS}} + \frac{N}{V_{OUT}}\right)$	4 A	4 A
Primary inductance L_{PRI}	$L_{PRI} = \frac{3\mu s \times V_{OUT}}{I_{PRI} \times N}$	11 µH	10 µH
NMOS: breakdown voltage V_{BR}	$V_{BR} \gg \left(V_{TRANs} + \frac{V_{OUT}}{N}\right)$ or $\left(L_{LEAKAGE}\frac{dI_D}{dT}\right)$	150 V	150 V
NMOS: average current I_{NMOS_AVG}	$I_{NMOS\_AVG} = \left(\frac{I_{PRI} \times V_{OUT}}{2 \times (V_{OUT} + N \times V_{TRANS})}\right)$	1 A	6.7 A
Output diode current I_{DIODE_AVG}	$I_{DIODE\_AVG} = \left(\frac{I_{PRI} \times V_{TRANS}}{2 \times (V_{OUT} + N \times V_{TRANS})}\right)$	0.1 A	4 A

Table 3-8 High voltage power supply functional parameter calculation

There are a number of components that need to be selected before carrying out topology specific modelling, simulation and the design of this CCPS. Correct component selection is important to achieve the specified performance. The design calculations and choice of components are discussed in detail in (Parthipan 2010) and have not been repeated in this thesis. However, it is summarised in Table 3-8 for completion. The design started with the selection of the V_{TRANS}, C_{bank+}, C_{bank-} , and P_{OUT_MAX}. These values were then used to select the functional parameters such as transformer turn ratio, N, the peak primary current, I_{pri}, and the primary inductance L_{pri} and appropriate components, such as the NMOS transistor (Q1), output diodes (D1 and d2) and other discrete components.

- Simulation and practical measurements

The HTPSU circuit was simulated for functional and power performance analysis. Component values are not included in the circuit for design confidentiality purposes. The simulation model for the controller IC was provided by the manufacturer and this can only be simulated in LTSpice. The rest of the components were modelled using basic SPICE models provided in the LTSpice model library. Simulation work carried out included the evaluation of the component selection and the HTPSU functional parameters and limitations.

Simulation data, showing a complete charging cycle of the capacitor bank, is presented in Figure 3-18; a. It shows the primary current I_{pri}, commutes through the NMOS transistor Q1 (hence $I_{d(Q1)}$) , power supply output voltages V_{out+}, and Vout- together with the Q1 gate drive signal V_{n013}. It is noticeable that the primary current $I_{d(Q1)}$ peaks as high as 26 A at start-up before settling down at 4 A, the value set for I_{pri} at the design calculation stage. This is acceptable due to

the short pulse duration and is handled by the input bulk capacitors (Figure 3-17; C3 and C6), not by the primary source – the battery.

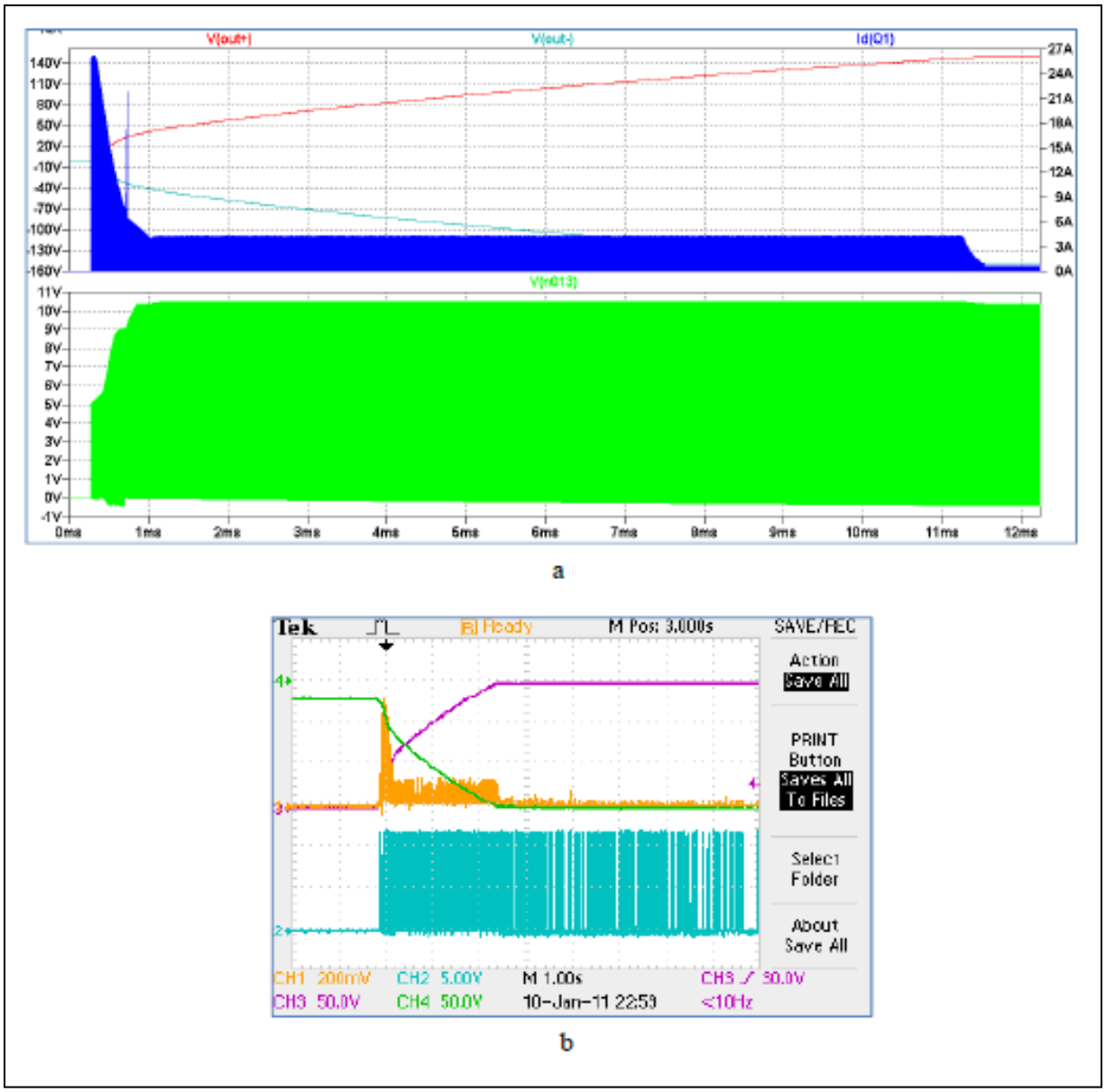


Figure 3-18 Operation of HTPSU (a. Simulation; b: Measurement). The charging current I_{pri} sourced from the battery (a: $I_{d(Q1)}$; b: CH1), switching signal (a: V_{n013}; b: CH2) and the charge voltages (a: V_{out+}, V_{out-}; b: CH3, 4) are shown

Measurement data obtained on the prototyped hardware is presented in Figure 3-18; b. With reference to it, CH1 shows the voltage across the current sense resistor (Figure 3-17;R4) of

value 27 mΩ, revealing $I_{d(Q1)}$, in other words the primary current I_{pri} peaks at about 18 A ($2.5 \times 200\, mV/R4$) at start-up, and drops down approximately to the set maximum value of I_{pri} of value 4 A ($0.5 \times 200\, mV/R4$). It can also be noticed that $I_{d(Q1)}$ is very minimal (~ 0.7 A) during the regulation phase, implying the power consumption is minimal at the regulation phase and highest in the charging phase.

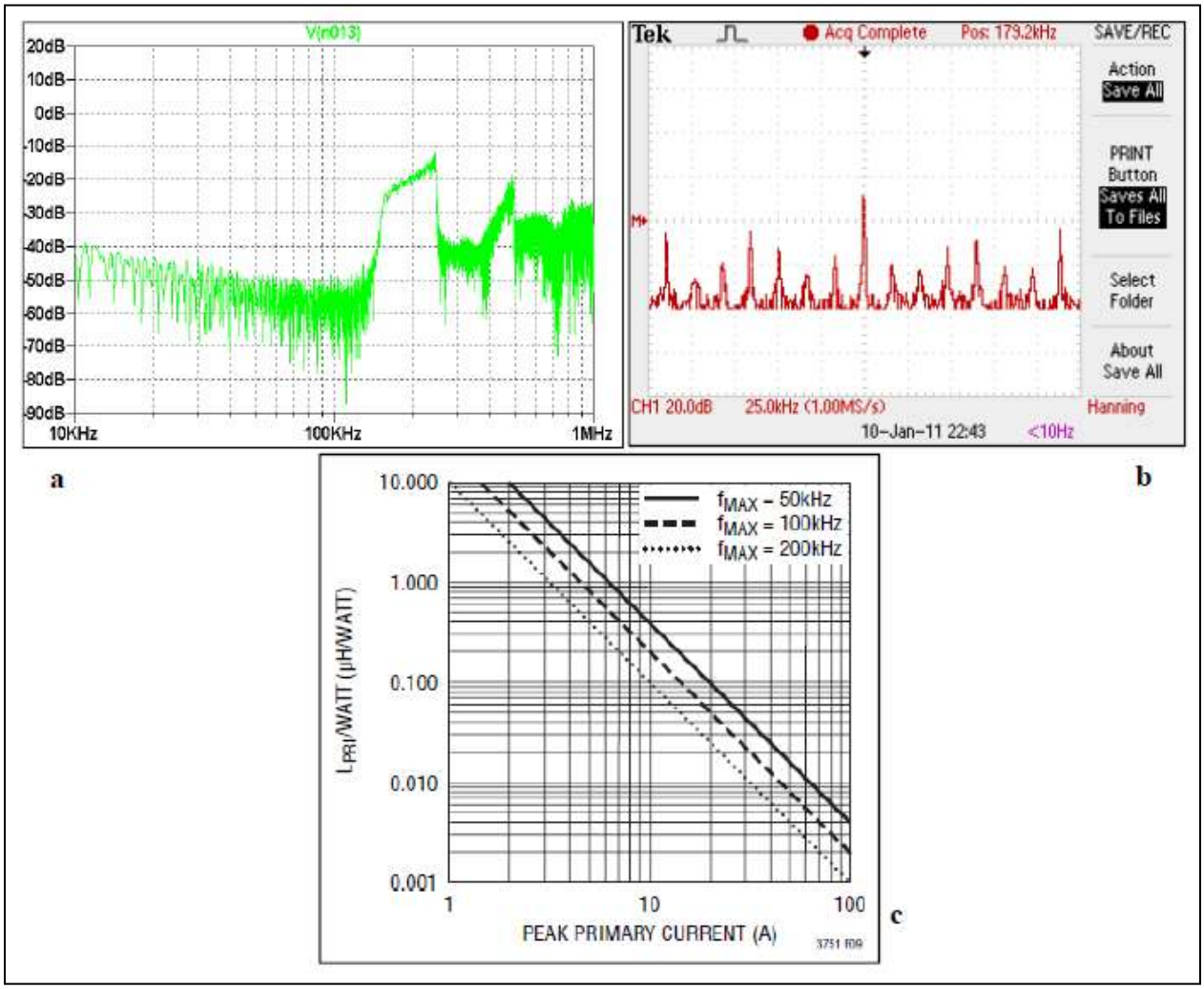


Figure 3-19 FFT of the switching signal (a: simulation; b: measurement) and guidance data from (LT3751 2008)

The gate drive signal of the switching device Q1 was recorded as V_{n013} (Figure 3-18; a) in simulation and CH2 (Figure 3-18; b) in measurement. It is noticeable that the voltage levels in

both cases are at 12 V for turning on and 0 V for turning off the power switch Q1. This matches the specified, output level of the gate drive pin of the controller IC (LT3751 2008). Moreover the FFT analysis on the gate drive signal reveals the maximum switching frequency, of in the region of 160 kHz - 200 kHz in simulation, and 179 kHz when measured in prototyped hardware. This is presented in Figure 3-19; a and b respectively for simulation and measurement. The evaluation of switching frequency is possible with the manufacturer's data published in (LT3751 2008). The published data is presented in Figure 3-19; c, showing the maximum switching frequency (f_{max}) expected for the design parameters; normalised L_{pri}/Watts and I_{pri}. These design parameters for this specific design are 10 μH/10 W and 4 A revealing the maximum switching frequency is around 150 kHz assuring the performance.

Excessive voltage and current stresses of switching components are a drawback to the use of flyback topology (Arash et al. 2008). The leakage inductance, of the transformer T1 leads to an excessive turn-off voltage at the power switch Q1. Allowing this voltage spike to exceed the breakdown voltage, V_{BR} of Q1 will degrade or damage it. Implementation of snubber circuits provides protection to switching devices by soft switching. However the extra circuits that implement soft switching cause power wastage, which is undesirable in energy aware applications. Moreover, the physical size also increases. In normal practice, designs allow space for the implementation of snubber circuits to be fitted if necessary. Simulations were carried out to see the effect of leakage inductance. The transformer manufacturers provided the value for the leakage inductance as 0.1 μH for the custom transformer manufactured for this design by transformer manufacturer Samuda as specified by (Parthipan T C8117 2010). This value was included in the simulation as $L_{leakage}$ in Figure 3-17.

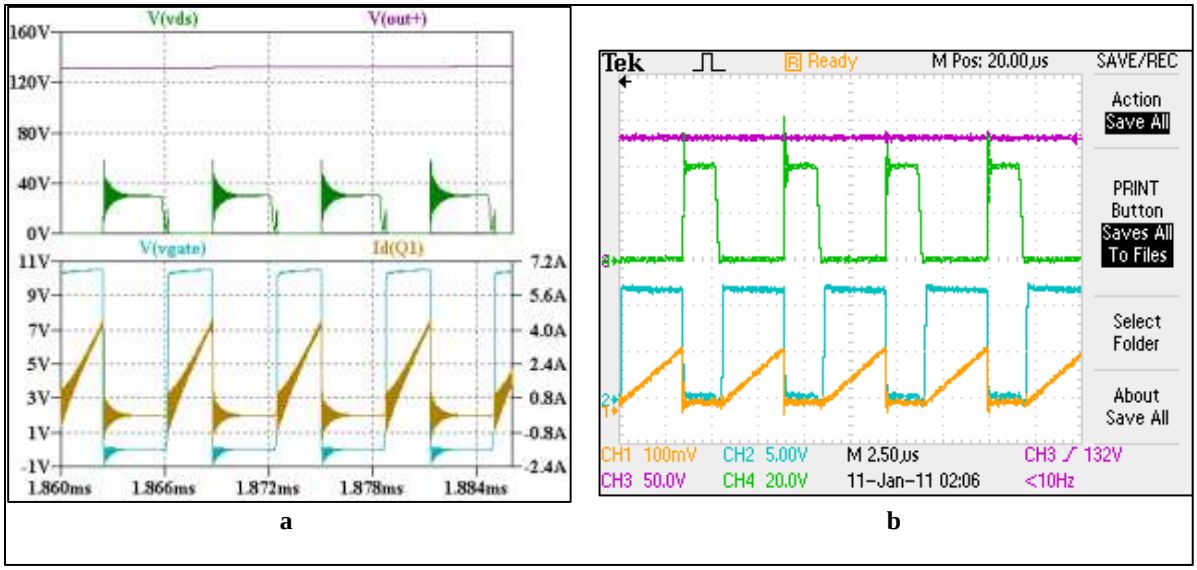


Figure 3-20 Boundary mode operation (BM) of the HTPSU (a: Simulation; b: Measurement). Voltage across the switching device V_{DS} (a: V_{VDS}; b: CH4) is as high as 65V due to $L_{leakage}$. BM of the HTPSU is noticeable as the drain current (a: $I_{d(Q1)}$; b: CH1) dies with the switching off of Q1 (a: V_{gate}; b: CH2)

The parasitic leakage inductance $L_{leakage}$ present in the design at switching off of the switching device Q1 caused a high voltage drop across the switching device (V_{DS}). Figure 3-20; a and b show the switching waveforms observed at the switching device terminals when a 4 A (a: $I_{d(Q1)}$; b: CH1) charging current was switched through this device in simulation and in real hardware respectively. The V_{DS} overshot to 60V in simulation (a: V_{ds}), which was close prediction to the hardware measurement of 65V (b: CH4). Both simulation and measurement data were recorded when the charge voltage V_{out+} was approximately 130V (a: V_{out+} and b: CH3). The stress on the switching device was observed to be high as the charging cycle approached the end of charging because of the high voltage and the high current experienced by the device. Careful layout of the components prevented V_{DS} overshooting above the V_{BR} of the switching device. This minimised the stray inductance around the circuit. Snubber circuits were not used in the design.

This circuit was designed to operate in boundary mode (BM) where optimal efficiency can be achieved in flyback topology (Basso 2008). In this mode the drain current ($I_{d(Q1)}$) drops to zero with the switching signal that turns on and off the device. This behaviour is noticeable in the simulation results presented in Figure 3-20; a, where $I_{d(Q1)}$ dies with the switching signal V_{gate}. This mode helps reduce transition losses occur during switching and allows efficient transfer of energy at switch Q1 opening. Measurement on real hardware presented in Figure 3-20; b – validates this as CH1 (the voltage across the current sense resistor (Figure 3-17;R4) of value 27 mΩ, revealing $I_{d(Q1)}$) reaching 0V with switching signal CH2.

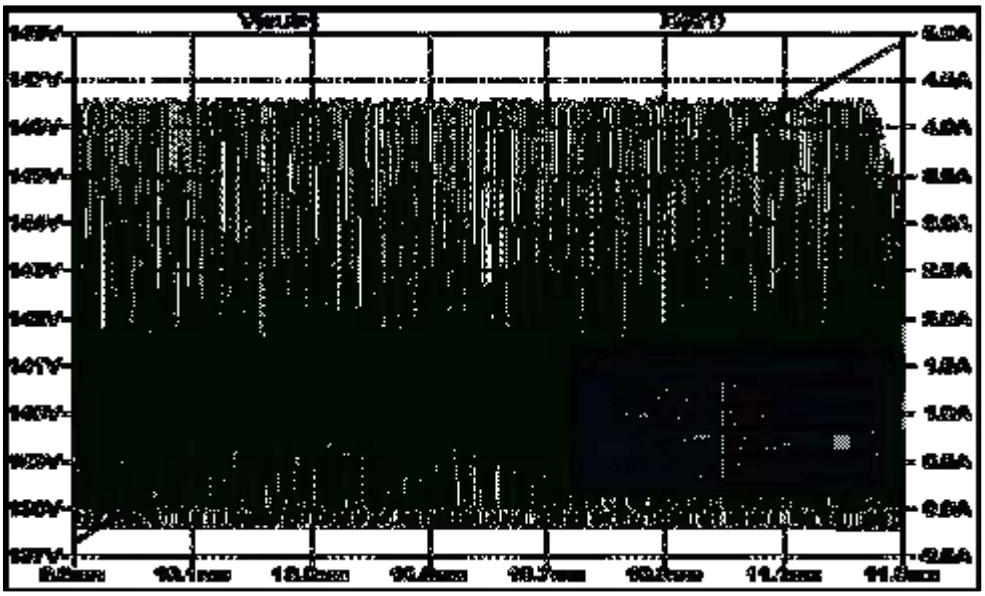


Figure 3-21 The average drain current (Id) passing through the switching device Q1 as the charging cycle is closer to the completion (simulation data).

The average primary current I_{NMOS_AVG} was calculated as 1 A (see Table 3-8), however in simulation it was found to be as high as 1.5 A as shown in Figure 3-21. The average current through Q1 is greatest as the output is nearing the full charge, due to a decrease in switching frequency at this stage. The higher rated MOSFET (150V; 6.7 A) chosen as a switching device in this design can handle the stress imposed by the high voltage level spikes of 65V and higher

current conduction. The output diodes were chosen as recommended by the manufacturer's datasheet (LT3751 2008) for its fast reverse recovery time (<100 ns) while satisfying the average current requirement.

- Loading effect on power supply

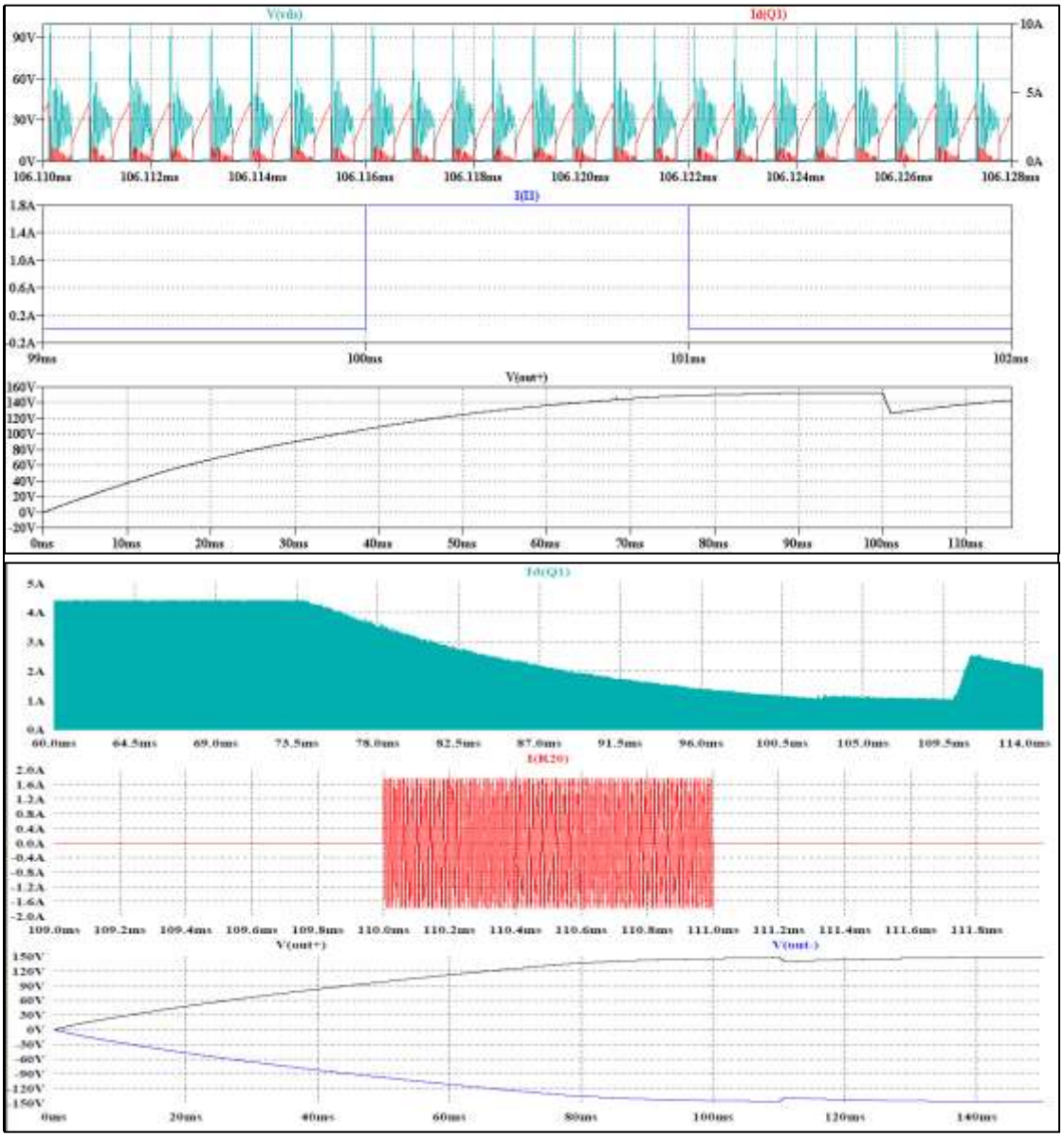


Figure 3-22 Loading of HTPSU with 1.8 A load current when V_{out+} and V_{out-} are charged to ±150V (top: square pulse; bottom: tone burst) in simulation. With a tone burst a voltage drop of 3V was experienced

The loading effect on the power supply output voltages when 1.8 A was drawn from it for 1 ms was simulated firstly with a square pulse as shown in Figure 3-22; top, and then with a tone burst of a sine wave of pulse duration 1 ms as shown in Figure 3-22; bottom. In both cases the drop in the output voltage rails was kept above 120 V; as required to satisfy the PSRR of the transmit circuit.

- Power Budget Analysis

The simulation results on power performance of the HTPSU are presented in Table 3-9. It includes the power dissipation across the main components together with the values for the power input and power delivered to the capacitor bank HT-Bank. An efficiency of 73 % is predictable from this data. This is comparable with the achievable efficiency of 70% for flyback topology (Christophe 2008) in general.

Description	Average Energy in 3 seconds
Energy drawn from the input voltage source E_{IN}	**273.7 W x 3 s = 821.1 J**
Power consumption of major components	
Controller IC	1.38 W
NMOS switching device	11.2 W
Output diodes	2.76 W
Sense resistor R4	11.29 W
Feedback resistor R5	2.35 W
Transformer T1	6.00 W
Other	4.00 W
Total power consumption by the circuit $P_{CONSUMPTION}$	**38.98 W**
Total energy consumed by the circuits $E_{CONSUMPTION}$	38.98 w x 3 s = 116.94 J
Energy delivered to capacitor bank	
C_{BANK+}	303 J
C_{BANK-}	303 J
Total energy delivered to capacitor bank E_{LOAD}	**606.0 J**
Energy unaccounted for ($E_{IN} – (E_{CONSUMPTION} + E_{LOAD})$)	32.72 x 3 = 98.16 J
Efficiency (100 * E_{LOAD}/E_{IN})	**73.8 %**

Table 3-9 Flyback High voltage power supply efficiency

3.5 Power budgeting and battery modelling

The operational time of the pulser-receiver unit is severely constrained by the available amount of energy in its primary power source – the Lithium-Ion battery. A battery based on Lithium-Ion (Li-Ion) chemistry was preferred in this application due to its high energy density compared to other rechargeable battery chemistries such as lead-acid, nickel-metal hydride and nickel-cadmium. The nominal open circuit terminal voltage of a single cell Li-Ion battery can vary depending on the cathode and anode materials used in the cell. This application used a Li-Ion battery aided with a lithium-nickel-manganese cobalt (NMC) oxide cathode and graphite anode, with open circuit nominal and maximum terminal voltages of 3.7 V and 4.2 V respectively. Maximum terminal voltages, in multiples of 4.2 V can be achieved by series connecting these cells.

Due to the complex electro-chemical reaction in a Li-Ion battery, its behaviour in terms of delivery of energy is non-linear and it can vary with the battery's age, discharge rate and the temperature that the cells are exposed to (Rao, Vrudhula & Rakhmatov 2003). Hence an adequate battery model is necessary to reliably predict the number of complete LRUT cycles that can be performed before the battery terminal voltage drops below the operational voltage.

Battery capacity is universally specified by the unit Amp-hour (Ah). Knowing the current drawn by the system for a particular functionality, and the discharge duration, allows one to compute the required energy capacity in Ah. Main functionalities of interest in one LRUT cycle are depicted symbolically in Figure 3-23. The description of the functions is tabulated in Table 3-10. Discharge current levels for each function were derived from the transient simulations of the subsystems, except for the digital circuitry. Datasheet values were used for computing the

digital circuits' power consumption. Note that the battery current level or function duration were not drawn to scale.

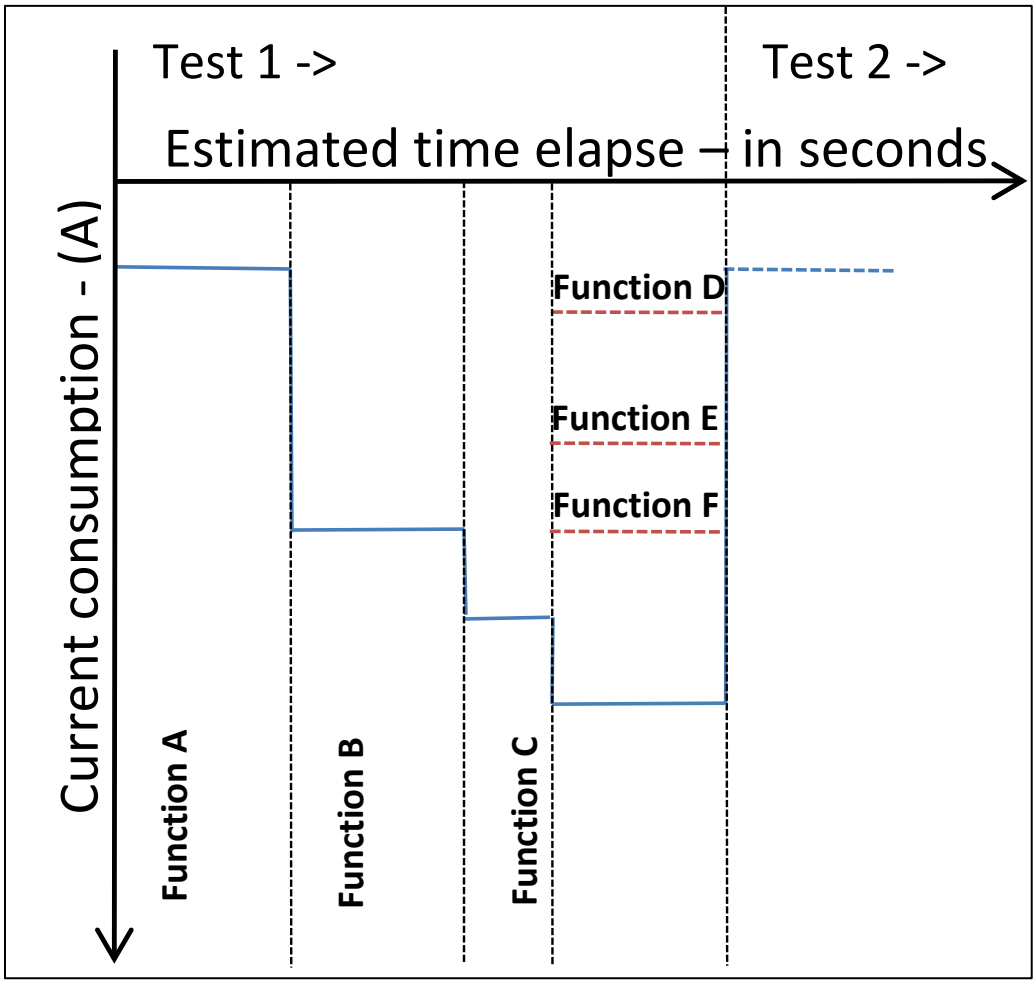


Figure 3-23 Sequence of battery current in one LRUT cycle

A fully charged battery in the pulser-receive unit is expected to last 12 complete LRUT cycles, referred to as test-1, test-2, etc., in Figure 3-23. Each LRUT cycle would step through activities from function A to F. Based on this, a power budgeting table was computed as shown in Table 3-11 and the required capacity of the battery that can adequately provide energy for 12 complete LRUT cycle was estimated as 6.5 Ah. The simulations and estimations were based on 15V power source.

Function	Description
Function A	Operator setup time. This can vary from operator to operator and location to location. Typical duration is 2 minutes, and the unit's standby current is estimated 300 mA based on quiescent current values of the major components.
Function B	Pumping of transducer array bladder, for appropriately clamping the transducers to the test specimen. This function lasts for approximately 2 minutes and the current draw is 3 A.
Function C	Charging of capacitor bank of value 1600 µF x 2. This function lasts for 3 s, and a maximum of 4A is likely to be drawn from the power source.
Function D	Once the capacitor bank is charged up, it is kept in regulator mode for the rest of the test which can be as long as 2 minutes. Simulations revealed the current drawn for this operation is about 0.75A.
Function E	Data collection; transmit and receive cycle repeated 16 times within one LRUT cycle in a typical application. Both activities are approximated to last 2 minutes simultaneously with function D. Receive activity draw about 2 A, based on 10 W power consumption and 5 V supply. Transmit activity during excitation of transducers, has been accounted for in function D. However the quiescent power consumption was not accounted for which can draw 10 mA per channel for the duration of 2 minutes. ADCs are on during this period, sampling the received data consuming 3W power. DACs are turned on for during this period, consuming 3W dynamic power for 1.1ms and static power of 0.1W (3.3V x 30 mA) for the rest of the period.
Function F	Digital electronics – this is considered static, and calculations based on datasheets extracted values, this function consumes about 10 W with a supply voltage 3.3 V, so 3 A is assumed to be drawn for this function for the complete LRUT cycle.

Table 3-10 function description

- Modelling of the battery

A suitable model for simulating this battery behaviour was researched to allow an estimation of the final battery terminal voltage after each LRUT cycle. Various models for Li-Ion batteries ranging from an analytical/mathematical model (Pedram & Wu. 1999), (Rakhmatov & Vrudhula 2001), (Rakhmatov 2005) to abstract models (Benini et al. 2000), (Kroeze & Krein 2008), (Gold 1997) have been cited in the literature. All these models had been critically reviewed by (Rao, Vrudhula & Rakhmatov 2003) highlighting the strengths and weakness of the models, together with their model descriptions. The Li-Ion battery model published by Gold (Gold 1997) is an electrical model, based on an E-Value ABM elements and discrete resistor and

capacitor models. This model was written in PSpice, which can be simulated in the LTSpice environment and allows simulating battery charging behaviour, discharge behaviour, battery life-cycle and temperature effects on the battery. This published work was claimed to be 88% accurate and the description of the model is very well documented in (Rao, Vrudhula & Rakhmatov 2003) and (Gold 1997). The model uses lookup tables which were formulated with values obtained by characterising a Li-Ion battery cell Polystor ICR018650, which has a design specification of 1.25 Ah with 4.2V (Max) 3.6 V (Nominal) and 2.75V (cut off) voltage with a cell series impedance of 80 mΩ.

Function	Description	Current consumption	Duration in seconds	As	As 24channels	Ah
Function A	Operator setup time	~300 mA	120 s x 12	432	-	120E-3
Function B	Collar inflation	~3 A	120 s x 12	4320	-	1.2
Function C	Initial High voltage bank charge up	4 A	3 s x 12	144	-	40E-3
Function D	High voltage bank regulation mode	0.75 A	120 s x 12	1080	-	300E-3
	Intermediate charging between averaging	4A	10 ms x 16 x 12	7.68	-	2.2E-3
Function E	Receive circuit (allowed 1s per shot)	10 W/5V = 2 A	16 s x 12	384	9216	2.56
	Transmit circuit standby	10 mA	120 s x 12	14.4	345.6	96E-3
	$ADC_{DYNAMIC}$ – assumed on throughout	140mW/3.3 = 42 mA	120 s x 12	60.5	1451.5	0.4
	$DAC_{DYNAMIC}$	120mW/3.3 = 36 mA	1.1ms x 16 x12	7.6E-3	0.183	0.05E-3
	DAC_{STATIC}	30 mA	120 s x 12	43.2	1036.8	288E-3
Function F	Digital electronics	12 W/3.3 V = 3.6 A	120 s x 12	5184	5184	1.44
Total	Approximated					6.5

Table 3-11 Power budgeting table

Figure 3-24 Equivalent circuit model of the battery pack resembling the prototyped battery pack. Each cell is an exact model of (Gold 1997) and parameterised to match Teletest battery pack. Load is provided by a current source.

A simulation model of the required battery pack was developed by arranging the simulation model of Li-Ion battery cell Polystor ICR018650 in 4 parallel by 4 series arrangement

as shown in Figure 3-24. Each cell is an exact model published by (Gold 1997) and parameterised to match prototyped/ specified battery pack. This arrangement in effect forms a simulation model for a 5 Ah, 14.4 V (nominal) battery pack. A decision to use an open circuit nominal terminal voltage of 15 V for the Li-ion battery was made, based upon the dimension of battery cells in the market that will fit the physical constraints of the prototyped product. A simple current source model (Figure 3-24; Discharge) was used for loading the battery pack during simulation. This model can be inputted with a stimulus data file, which represents the battery discharge current data recorded during measurement or compiled using data obtained by simulating the system constructs. This stimulus data resembles the sequence of the battery discharge current pattern shown in Figure 3-23 for one LRUT cycle. Usage of this model avoids the need of including all simulation models developed for simulating the loading effect on the battery, hence simplifying modelling, which speeds up simulation.

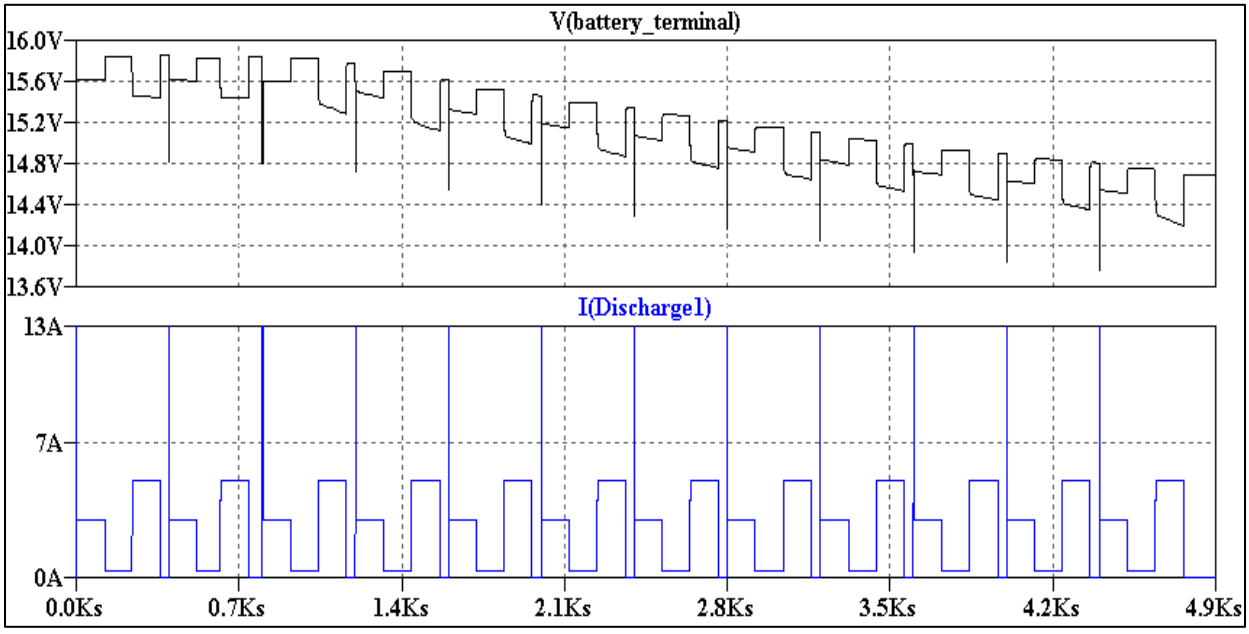


Figure 3-25 Imitated discharge current loading the battery pack simulation model

Figure 3-25; top, shows in simulation, how the terminal voltage of the developed battery model drops with discharging when it was loaded with repeated discharge data. Here the discharge data input was compiled using the data obtained from previous simulations and predictions. The same discharge cycle/pattern was repeated 12 times as shown in Figure 3-25; bottom, where the pattern of the $I_{Discharge1}$ waveform is periodic of the LRUT cycle, having attributed as Figure 3-23. It is noticeable from Figure 3-25; top, the battery model showed a terminal voltage of approximately 15.8 V initially and after 12 LRUT discharge cycles, it dropped down to a lower value of approximately 14.6 V. The high 13 A current spike at the beginning of each LRUT cycle is expected to occur every time the pump that inflates the transducer array is started (see section 2.3). This high spike in reality lasts for 100 ms (not visible due to the high time base setting on the plot) and pulls the battery terminal voltage down which is the likely reason for tripping/collapsing the battery terminal voltage. This is also noticeable on the graph.

- Evaluation of the model

Evaluation of the battery model was carried out by firstly recording the current draw/discharge trend when the prototyped battery pack was loaded with the real hardware of a prototyped pulser-receiver, and then the recorded data was inputted in the simulation via a current source shown in Figure 3-24; discharge1, so that the developed simulation model is loaded with the real measurement data instead of the compiled data used in the previous simulation. The results are presented in Figure 3-26, showing the final battery terminal voltage dropping down to approximately 14.8 V after 12 repeated LRUT cycles. This final battery terminal voltage is in reality higher than the theoretical minimum battery terminal voltage 13.3 V

possible according to the cell manufacturers. Hence, the simulation was extended to foresee the maximum complete LRUT cycles possible before the battery collapses.

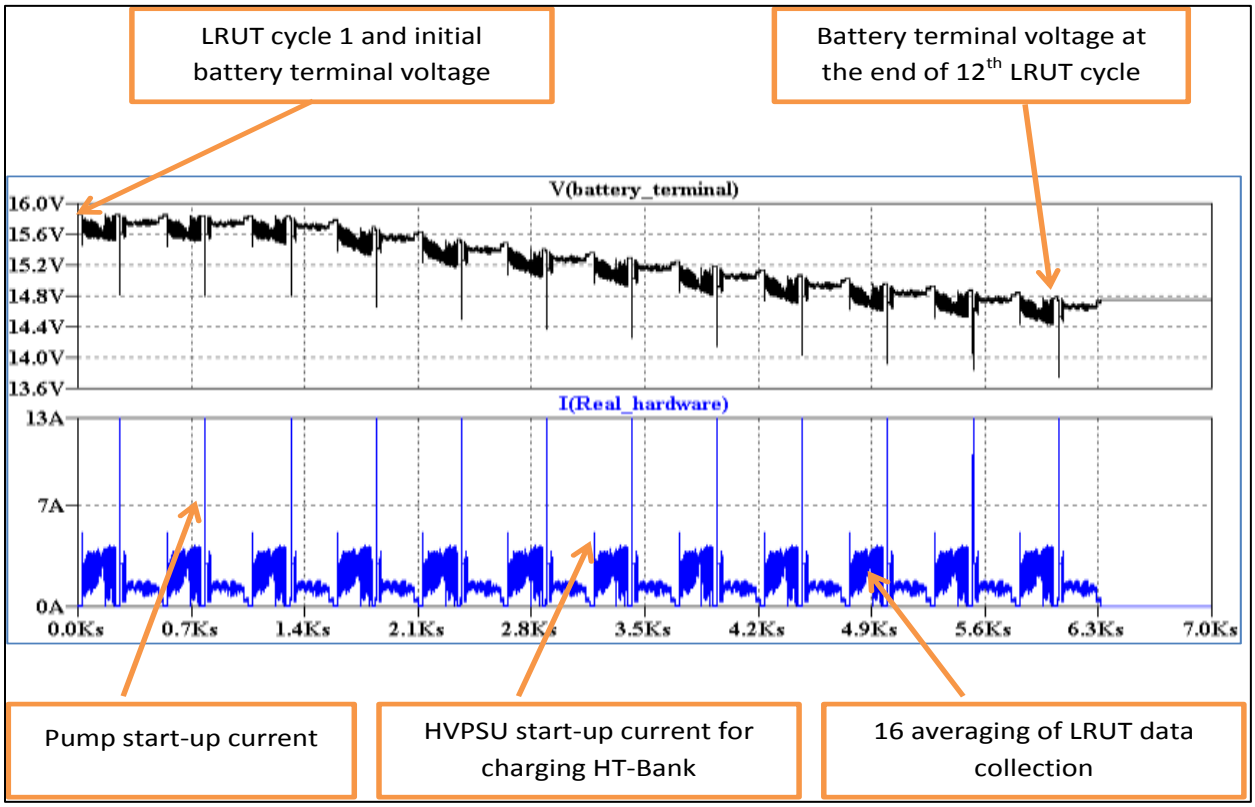


Figure 3-26 Real hardware loading effect on the Li-Ion battery simulation model

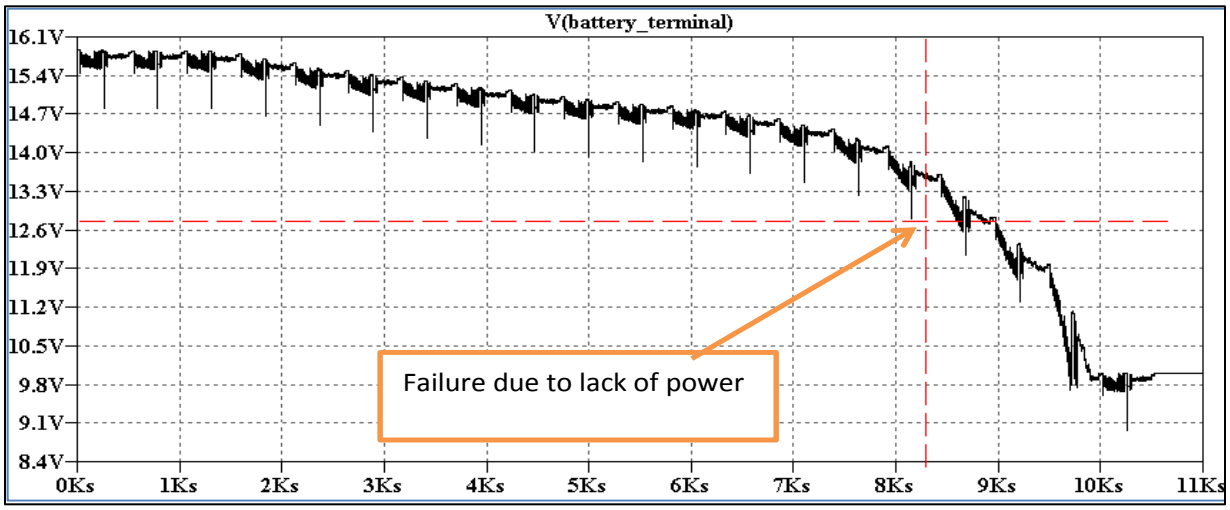


Figure 3-27 Number of completed LRUT cycles in simulation

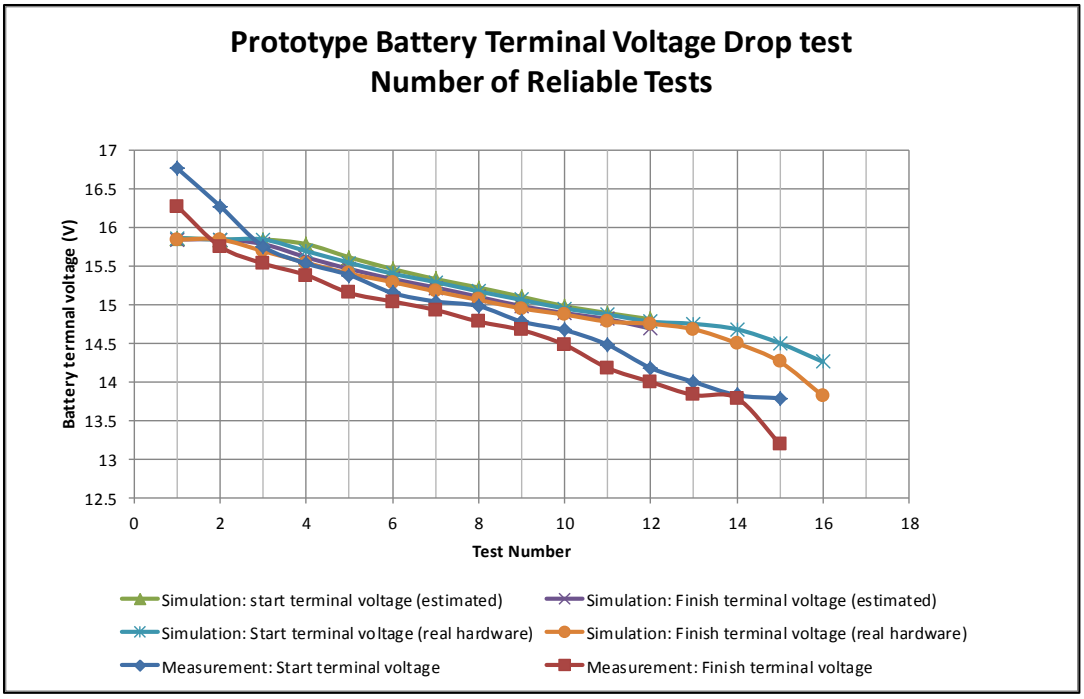


Figure 3-28 Number of completed LRUT cycles and battery terminal voltage drop

Extended simulations carried out had the number of LRUT cycles extended to more than 12, as shown in Figure 3-27. It shows the possibility of executing 16 complete LRUT cycles before the battery terminal voltage terminates/collapses. This is indicated by the red-dashed lines. In reality 15 shots were achieved with the prototyped hardware and a fully charged, brand new battery. The measurement results together with the simulation data are presented in Figure 3-28.

3.6 Hardware realisation

The hardware discussed was prototyped, and eventually has been launched as a commercial product. The prototype development of the power electronics circuits and the primary power source – the battery for the pulser-receiver unit used system modelling techniques for the design process. It allowed prototypes to be produced with very minimal resources used,

avoided over-specified component selection and reduced design iterations. This section includes some of the traces obtained during the prototype development.

3.6.1 PSU prototype

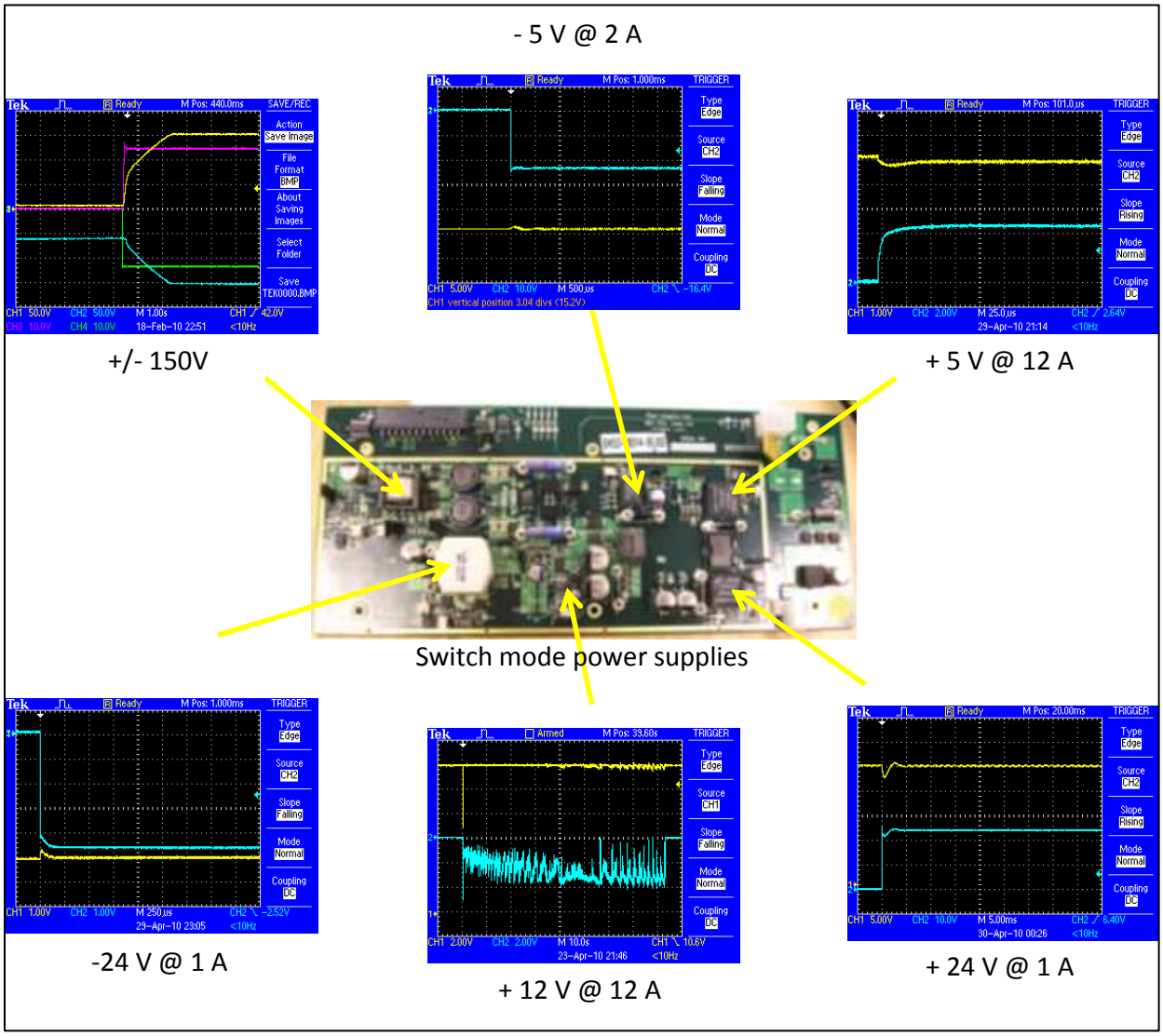


Figure 3-29 Power electronics hardware for pulser-receiver unit

The power electronics circuits of the pulser-receiver were designed to produce power levels + 5 V; 12 A, -5 V; 2 A, +24 V; 1 A, -24 V; 1 A, 12 V; 12 A and a ±150 V high voltage power supply. Figure 3-29, shows the power supply hardware that generates all these power

levels from a 15 V (Nom); 5.3 Ah Li-Ion battery, with the traces captured during the prototype testing of all the mentioned power supply domains. The traces show their corresponding output voltage level while drawing the maximum current that each power supply was designed to handle. The hardware performed satisfactorily and did not require design iteration.

The prediction of the HT-Bank capacitance value and the battery capacity during the study allowed the prototype to be designed in a compact form as shown in Figure 3-30, which allowed the pulser-receiver unit to be designed with a one third reduction in weight and size without compromising the performance or operational length of time. Note that the capacitance value for HT-Bank is the same for the previous version of the pulser-receiver but the research work and the system level simulation allowed confirmation of its value which was impossible otherwise.

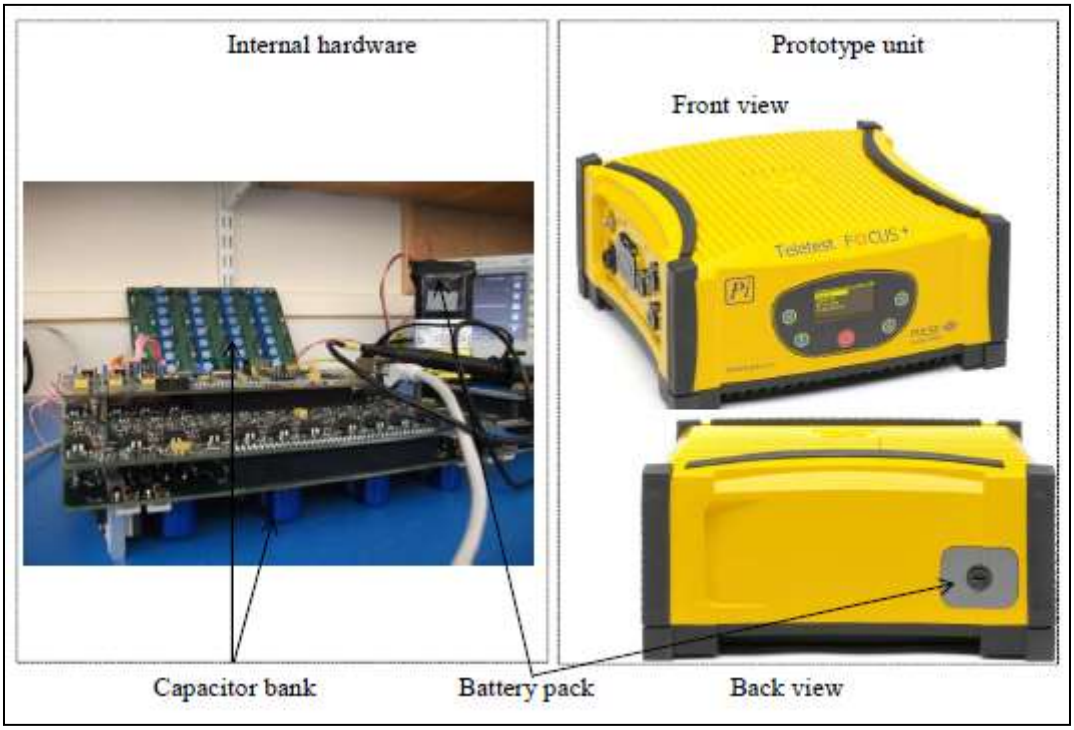


Figure 3-30 HT-Bank together with its associated transmit and receive circuit

Figure 3-31 Pulser-receiver unit as a commercial product

A commercial product was manufactured based on the prototype. Figure 3-31; left and right, shows the size comparison between the latest and the previous version of the pulser-receiver units. Practical tests carried out on the manufactured pulser-receiver unit revealed a fully charged battery of specification 15 V (Nom); 5.3 Ah allows 20 complete inspection cycles. This satisfies the predicted performance of 12 inspection cycles stated above. An increase in the number of LRUT cycles in the production version was due to the enhanced pump assembly and the reduction in standby/ setup time achieved by means of the software enhancement. The new pump assembly only takes about half the time to inflate the collar compared to the time taken by the previous pump assembly and the predicted 2 minutes in the calculations. Software enhancement increased the speed of transferring data to the PC, and provided an easy to use graphical interface for rapid setup and configuration.

3.7 Summary

This Chapter has described building a custom simulation tool box that can be used in the design process for product enhancement. Once it was built it was then fully utilised to analyse

power and functional performance of the system against industrial requirements. The goal of enhanced portability was achieved, with efficient usage of resources.

The choice of model topology and relaxing of simulation settings were found to be a valuable tactic in the development and simulation of this tool set for achieving acceptable accuracy and to overcome non-convergence issues experienced especially because of its non-linear circuits. Full-featured topology specific models were found to be accurate but difficult to simulate. On the other hand, equivalent circuit models were also found to be equally useful and produced acceptable simulation results, despite their lower accuracy. However, the accuracy of the topology specific models, especially those used in the modelling of the power supply circuits and the transmit circuits, provided a powerful tool for debugging and observing component specific issues at the prototype and maintenance phases, and subsequent prototyping of the circuits generated from the modelling work.

The research enabled the instrumentation utilised in the LRUT application to be assessed for its power performance for various load conditions and variants of hardware configurations. This work made it possible to reliably specify a suitable primary power source for PI's flagship product Teletest®, well before the hardware had been developed. To produce sensible simulation results that can be utilised in the process of evaluating the portability enhancement the following assumptions and compromises were made:

1. Model evaluation against measurement data at component level was minimal. Obtaining measurement data for validation at a component level suffered, because of limited resources and unavailability of hardware at the early stages. Instead, expert knowledge in the form of measurement data obtained from previous hardware in separate work,

subsystems' specifications and theoretical calculations were used for model evaluation. This was accepted as adequate for two major reasons; first most models used in the development of all power sensitive sub circuits were topology specific models and were assumed accurate, second the equivalent circuit models developed for two main components – the battery and the transducer, which strongly influenced the decision making, were validated against measurement data. The prototyped power supply designs were also validated using measurement data.

2. Downsizing the primary power source – the battery was identified to be the only possible option for enhancing portability of this system. Computation of power consumption data of the system was required for allocating a suitable power source. It assumed a worst case scenario on every single active cycle of the system for computation of the average power computation. This was conservative to account for possible inaccuracies in the model development and any human error possible in operation.
3. A smaller size and less weight of the Li-Ion battery pack were only possible with a lower battery capacity. Capacity reduction introduced potential risks of shortening operational time and functional performance, by limiting the battery cells' ability to handle a high rate of change of discharge current. In essence, it was the battery terminal voltage that needed to be maintained above the required minimum operational voltage of the system, for any possible discharge current rate the system imposed on the battery. With the available battery cells in the market it was possible to stack a number of standard cells in a serial-parallel combination to achieve the required terminal voltage and capacity.
4. A model accuracy of 20% was accepted for the transducer equivalent circuit models and efforts were made to produce simulation data that agreed with the measurement data

within the application range. These attributes somewhat limit this model's ability to adapt or assess hardware for future application enhancements. The system modelling rule of including only what is needed in the simulation model was followed to guarantee success of achieving the goal on time. The models produced are suitable for immediate future applications regardless. Further work on refinement of the equivalent circuit model of the piezoelectric element, and the Teletest transducer is recommended for ease of parameterisation and gaining confidence on its accuracy. The same argument applies to the battery model used; although the data used in the model were not produced for the exact battery cell used in the real hardware, the model provided a close prediction. The LRUT instrumentation is likely to use the chosen battery cell with variants of combinations in its future designs. Hence characterisation of the very battery cell used in the custom battery made is encouraged for the refinement of the model.

5. Simultaneous simulation of all developed models in a single simulation platform was not performed to avoid complexity and improve simulation time. This simplification was assumed acceptable, because the subsystems that affected the battery behaviour differently when connected to another subsystem were simulated simultaneously with other relevant subsystems to capture the difference. Simulating the drive circuits with a transducer array allowed analysing and producing data on these circuits' dependency on load behaviour. The utilisation of this data in the specification of the high voltage power supply for the drive circuits (HTPSU) has proved essential to maintain PSRR of these application critical circuits.
6. The modelling work explored the possibility of including a feature of analysing the port dynamics of the LRUT instrumentation by closing the loop between the transmit and

receive sides of the system, by the inclusion of the electro-acoustic nature of the medium. This is discussed in Chapter 4. It was appreciated that it is almost impossible to model and simulate the acoustic nature of the medium on which the LRUT technique is applied in PSpice, however efforts were made to demonstrate the ability of the developed model to adapt to include abstract information of components, with the functionality to model coupling between two or more engineering disciplines.

The outcome of this part of the research is three fold; firstly, an adequate battery that allows reliable LRUT inspection for the planned period of 12 hours/ 12 tests, while improving the portability by 33% in size and weight, secondly the process allowed rapid prototyping of LRUT hardware sustainably and thirdly, the model provided a platform on which variants of this hardware can be assessed with no concern of damaging or utilising useful resources. However, this work has identified the need for a platform free template with all necessary formulae to provide information on battery capacity requirements for variants of hardware configuration. This would benefit the industry, where expert knowledge is not always available to use the simulation tool. Initial work on this had been demonstrated with an Excel based template, where the built-in formulae were obtained, using a curve-fitting method on the obtained power performance graphs produced from simulation data.

4 MEDIUM FREQUENCY LRUT APPLICATION – SYSTEM MODEL REUSE

4.1 Overview

One of the efficient features of using a system modelling technique is the ability to reuse the developed system model for verifying and/or studying new concepts and to thereby accelerate the design process with minimal resource usage. This Chapter introduces one such application in which the system models that were developed for power performance analysis in Chapter 3 were redeployed for evolving the existing low frequency LRUT process and hardware to its next phase of a medium frequency application for inspecting complex structures. This design process required rapid prototyping to prove the concept.

4.2 Medium frequency LRUT application

LRUT of complex structures such as aircraft components is challenging as the reflection from the features becomes more complex and harder to analyse. This research programme looked at inspecting one such complex component used in the aircraft wing assembly, which experiences high and variable stresses in operation, leading to potential fatigue cracking, but has limited access for inspection and maintenance. This component is graphically shown in Figure 4-1 with possible locations for mounting transducers for LRUT inspection using pulse-echo or pitch-catch methods. The Finite Element Analysis (FEA) carried out and published by (Haig & Stavrou 2012) reports, for reliably detecting flaws in these type of complex structures, the LRUT technique utilising an ultrasonic wave frequency in the high 100's of kilohertz is required, in contrast to the low frequency LRUT application employed in pipeline inspection.

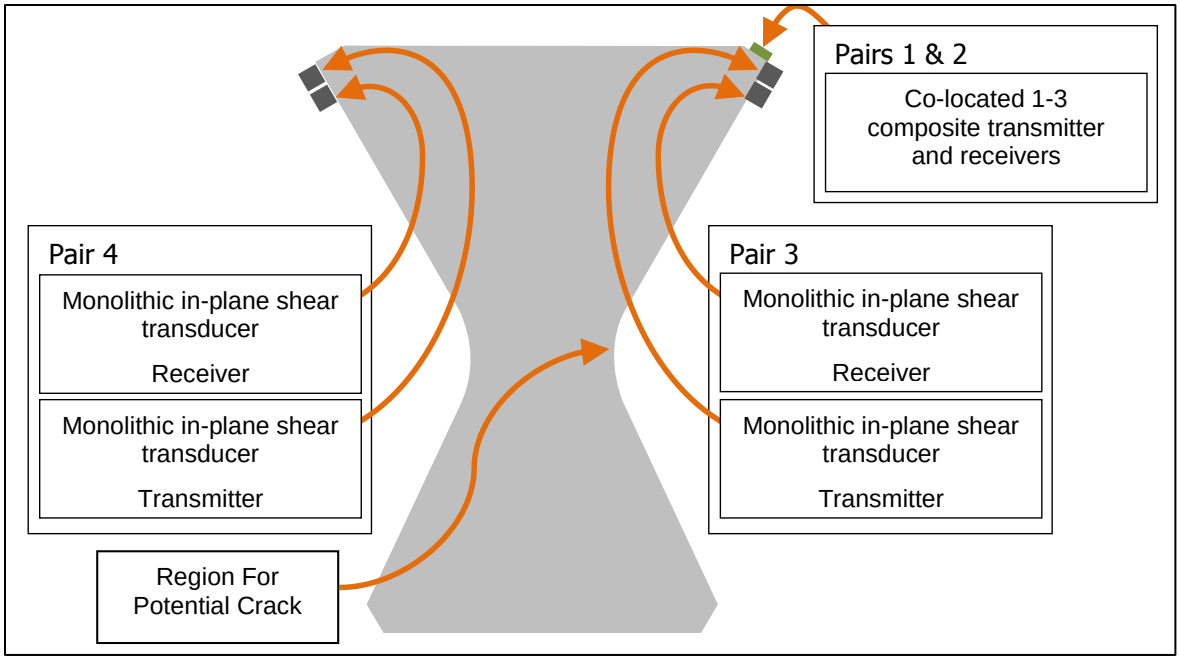


Figure 4-1 Complex structure of an aircraft wing assembly component (Haig & Stavrou 2012)

4.3 Reusability of existing hardware

The complex features of the aircraft components require a medium frequency ultrasonic wave-mode in the frequency range of 300 kHz and 1 MHz for inspection as (Haig & Stavrou 2012) proved using FEA modelling. The transmit and receive circuit in the pulser-receiver unit discussed in Chapter 3; section 3.3.1 and section 3.3.3, respectively were designed for low frequency applications and cannot handle the higher frequencies proposed by (Haig & Stavrou 2012) for experimental work i.e. in the range of up to 1 MHz. The project required rapid prototyping of a medium frequency instrument.

The existing hardware, discussed in Chapter 3 was assessed first and then the feasibility of modifying its circuits to serve this purpose was analysed. This was carried out by means of simulation using the developed LRUT system models discussed in Chapter 3. Once suitable component and/or circuit changes were identified the corresponding components in the

developed simulation models were replaced with the new models and simulated for validating component and/or design changes.

4.4 Development of high frequency transmit circuits

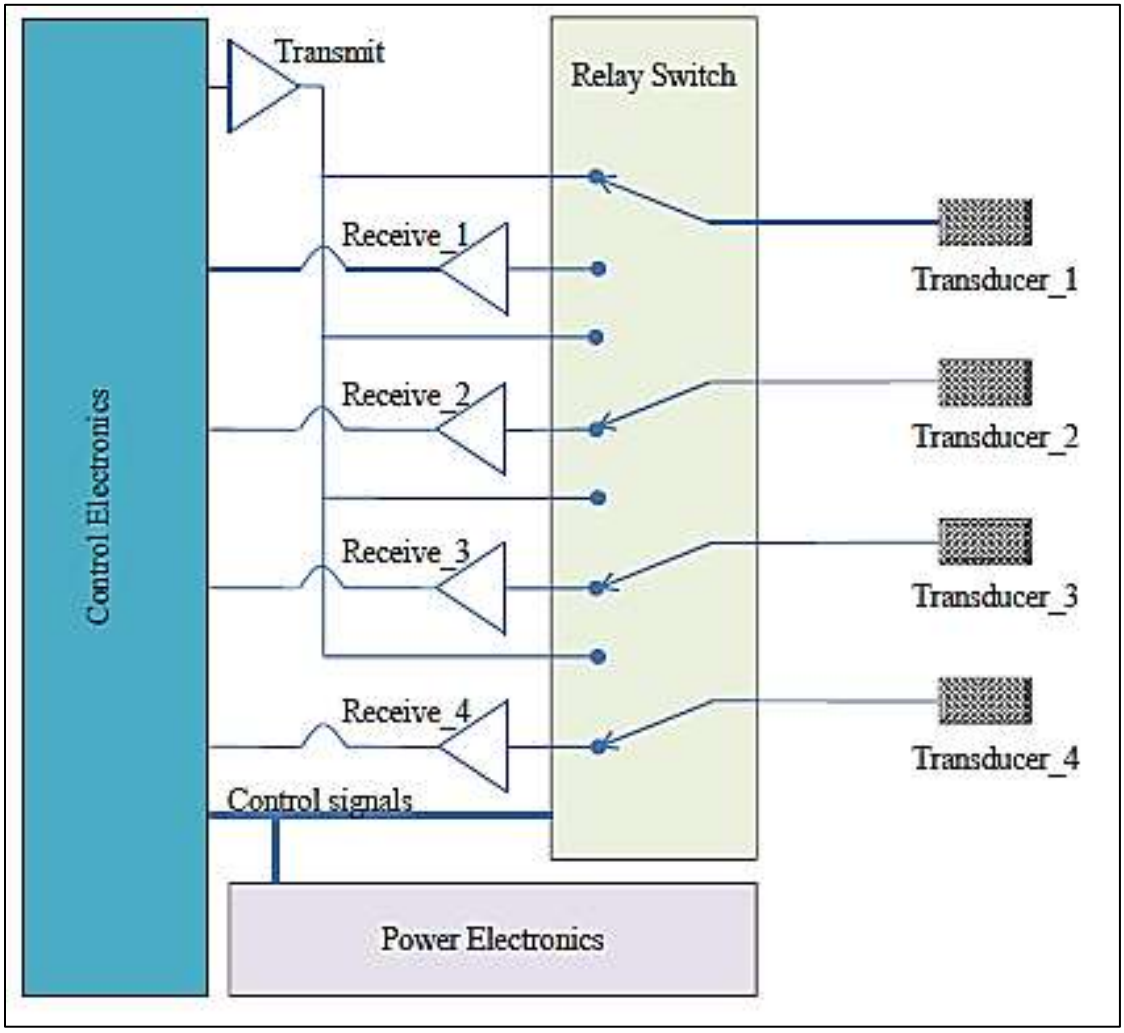


Figure 4-2 Medium frequency LRUT hardware architecture

The proposed hardware for the medium frequency LRUT experiment is shown in Figure 4-2, where a single transmit channel is multiplexed to four individual LRUT-transducers using DPST (Dual Pole Single Throw) relay switches. The control electronics switches the relays in the following manner; the transmit circuit excites one of these transducers while the other three

transducers are switched to three of the four receive circuits, receiving the reflected signal from features such as flaws and edges of the test specimen.

The LRUT-transducers discussed in Chapter 3; section 3.2.2, were used. Rapid prototyping was carried out by firstly simulating the concept in the LTSpice computer environment utilising previously developed LRUT system models with suitable modifications to transmit, receive and the power supply electronics. The digital logic for control, waveform generation and data storage was facilitated using commercially available data acquisition instrumentation from scientific hardware manufacturer National Instruments.

4.4.1 Transducer model and its performance

This particular application requires a single LRUT-transducer to be excited with an electrical excitation signal with a frequency in the range of 10 kHz and 1 MHz and with a maximum amplitude of ±80V. The input impedance analysis carried out reveals the drive electronics is likely to drive a load impedance of 148Ω at 1 MHz, which is equivalent to 1.3 nF in capacitance value at 1 MHz, considering 20% tolerance on the LRUT-transducer impedance.

The impedance characteristics of the LRUT-transducer, showing the impedance values for the extended frequency application of 1 MHz, are shown in Figure 4-3. Both simulation and measurement results obtained using an impedance analyser are presented with error bars set on each experimental data point. Error bar setting on the impedance curve and the phase curve are set to ± 20% and ± 2% respectively. Simulation results deviate from the measurement results by nearly 40 % at the high frequency end. Optimisation of the model was not considered for the following reasons; the model accuracy was proved to be acceptable in the previous application

discussed in Chapter 3 and the impedance values for 80% of the frequency range of interest agree with the measurement data with moderate accuracy of around 30% closer to the end.

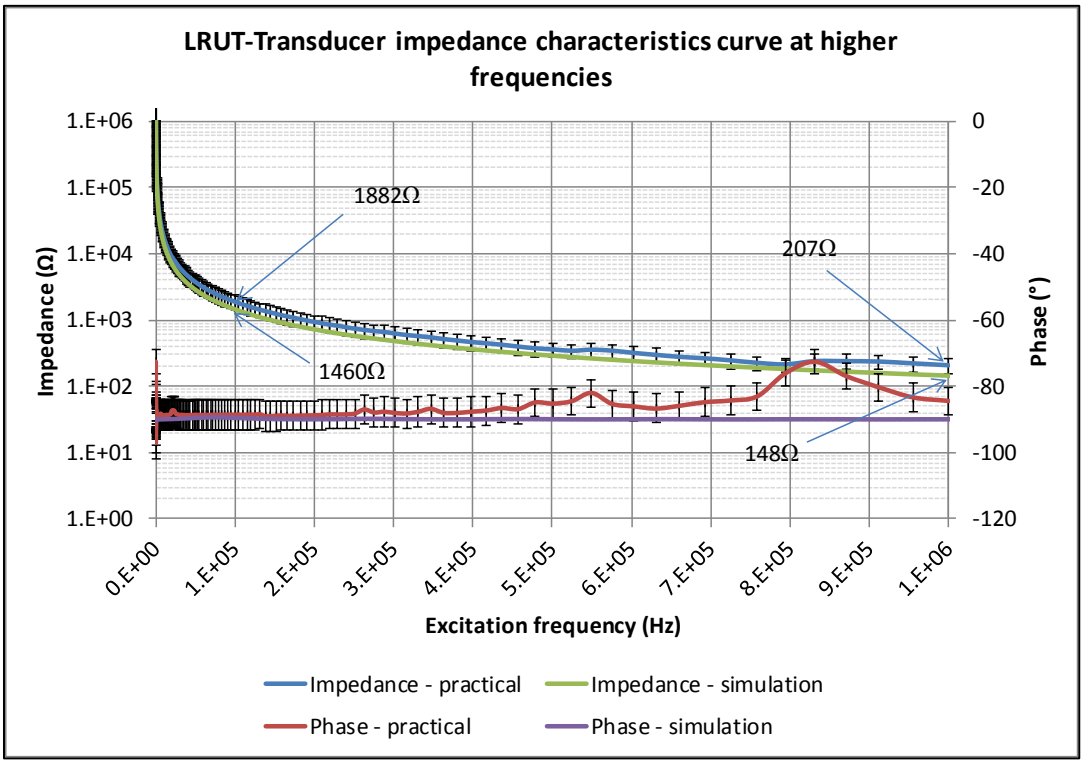


Figure 4-3 Impedance characteristics of an LRUT transducer for extended 1 MHz application

An appropriate drive circuit (transmit) was required to sufficiently excite an LRUT-a transducer for the frequency range 20 kHz – 1 MHz at ± 80V.

4.4.2 Medium frequency transmit circuit

Parameter	Symbol	Value
Supply voltage	V_S	$\pm 100V_{DC}$
Excitation signal frequency	F_{EXT}	Min = 20 kHz; Max = 1 MHz
Excitation voltage	V_{EXT}	± 80 V (Peak ≅ 80 V)
Load	C_{LOAD}	Maximum of 1 LRUT-transducers ~ 1.3 nF
Input to transmit circuit	V_{IN}	Von-Hann windowed Sine wave (20 kHz to 1 MHz)
Voltage gain	G_{PA}	Adjustable – default 27 dB

Table 4-1 Medium frequency transmit circuit specification

The specification of a transmit circuit for facilitating medium frequency LRUT is given in Table 4-1. Design specific calculations are included in Table 4-2 for component selection and evaluation (Cirrus logic 2009).

Parameter	Formula	Value
Slew Rate (SR)	$SR = 2 \times \pi \times F_{EXT} \times V_{EXT\_MAX} \times 1 \times 10^{-6}$	630V/µs
Total load Impedance (X_C)	$X_C = 1/(2 \times \pi \times F_{EXCIT_{MAX}} \times C_{LOAD})$	148 Ω ± 20%
Peak load current (I_{PEAK})	$I_{PEAK} = SR \times C_{LOAD}$	0.8 A
Worst case power dissipation (P_{TX_WORST})	$P_{TX\_WORST} = 4 \times V_S^2/2 \times \pi \times X_C$	53 W

Table 4-2 Medium frequency transmit circuit calculations

Satisfying the required functional specification to sufficiently excite the LRUT transducer at 1 MHz cannot be met by the transmit circuit discussed in Chapter 3; section 3.3.1, because of the slow slew-rate of the power amplifier used in the circuit (300 V/µs), despite its ability to comfortably satisfy output voltage and load current requirement. The LTSpice simulation model developed for the transmit circuit previously, allowed individual components to be evaluated for their performance when integrated in the circuit design, due to the topology specific modelling methodology followed in the model development of this circuit.

The slew-rate limited power amplifier model in the existing transmit circuit (Chapter 3; section 3.3.1) was replaced with a suitable power amplifier model with necessary modification in the circuit and simulated for the circuit and component functional performance. The circuit diagram is similar to the simplified circuit shown in figure 3-8, which shows the off-the-shelf power amplifier and the complementary MOSFET circuit with over-current protection. The power amplifier used is capable of delivering 5 A pulsed current. However, the complementary MOSFET circuit was left in the circuit as a buffer to protect the power amplifier, should a short circuit occur in the load. The power amplifier utilised does not have integrated overcurrent protection circuit.

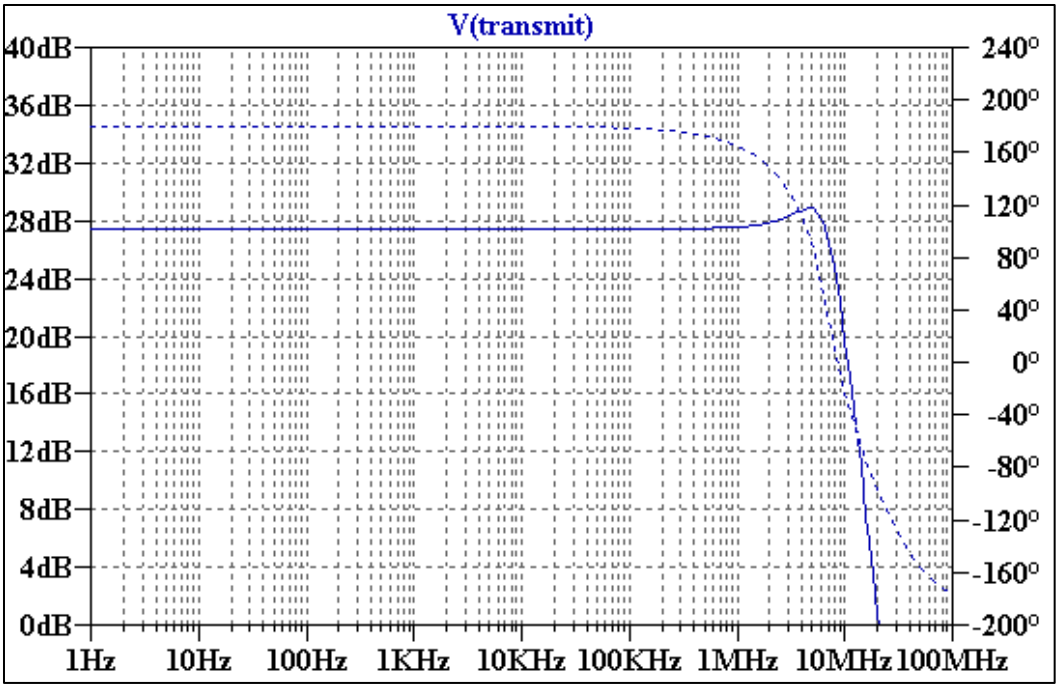


Figure 4-4 Bode plot of the medium frequency transmit circuit in simulation

Both AC and transient analysis were carried out to make sure an undistorted performance is achieved. Figure 4-4 shows the frequency response of the modified transmit circuit, depicting its ability of handling the application specific frequency range of up to 1 MHz without distortion. A set gain of approximately 27 dB was maintained for the entire frequency range. The phase value of 180° represents the inverting amplifier configuration.

Transient analysis at the output of the modified transmit circuit model was obtained by simulating the model with an input signal of amplitude ± 4.3V and frequency 1 MHz. With this input amplitude and the gain set to 27 dB, an output of 96V is achievable at the output of the transmit circuit (ignoring voltage drop across devices). An undistorted excitation signal of approximate amplitude of ±90V and frequency 1 MHz was observed in simulation as shown in Figure 4-5; a. A voltage drop of approximately 7V was noticed across the complementary MOSFET pair as expected. The output current delivered to the load was observed to be as high

as 700 mA. In both frequency and transient analysis simulations the LRUT-transducer model discussed in Chapter 3; section 3.2.2 was used as a load.

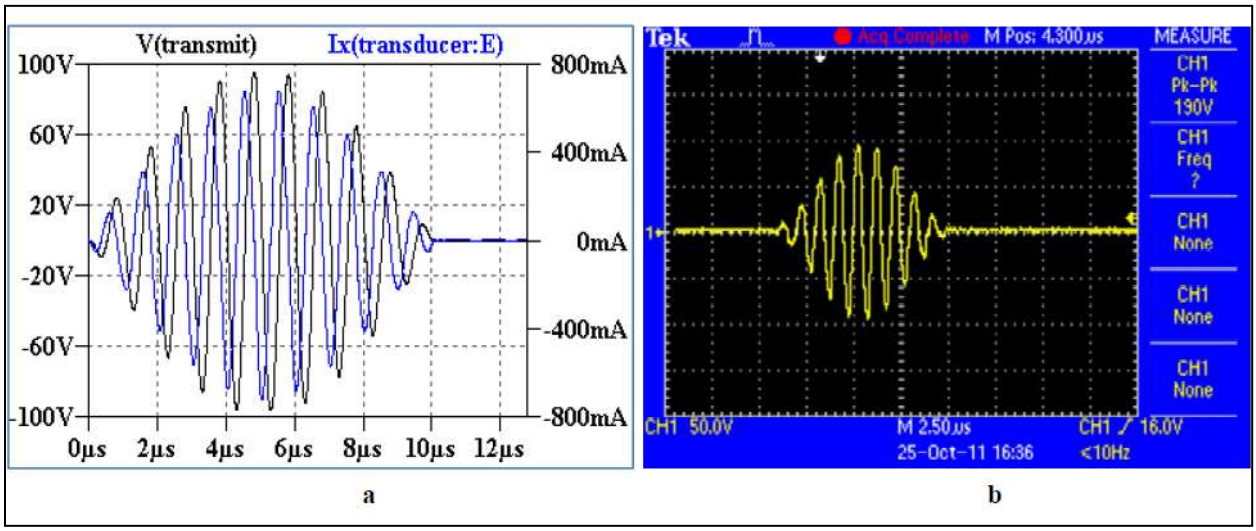


Figure 4-5 Excitation signal driving LRUT-Transducer showing 190$V_{pk\text{-}pk}$; at 1 MHz (a: simulation; b: measurement)

The circuit was prototyped and the output waveform observed, as shown in Figure 4-5; b. It is noticeable that this waveform matches the simulation results; amplitude ±90 V and frequency 1 MHz.

4.4.3 Medium frequency receive circuit

The receive circuit requirements for this application were specified to have a selectable gain of 20 dB or 40 dB, with a bandpass filter having cut-off frequencies of 150 kHz and 1.2 MHz, so that a flat response can be achieved for the frequency range of interest between 300 kHz and 800 kHz as reported by (Haig & Stavrou 2012). The receive circuit discussed in Chapter 3; section 3.3.3, is not adequate to facilitate the required high bandwidth specification as it was designed to handle input signal frequencies of up to 100 kHz for a typical LRUT application on pipes. The simulation model developed for the receive circuit was modified with appropriate

components and simulated before prototyping. The frequency response obtained in simulation (shown in Figure 4-6) reveals the ability of the receive circuit to meet the specification of approximately 37 dB gain and cut-off frequencies for high-pass and low-pass filters at 150 kHz and 1.2 MHz respectively. This simulation was setup with a gain setting of 40 dB and the reduction in amplification (~ 3dB) was due to the passive filter arrangements.

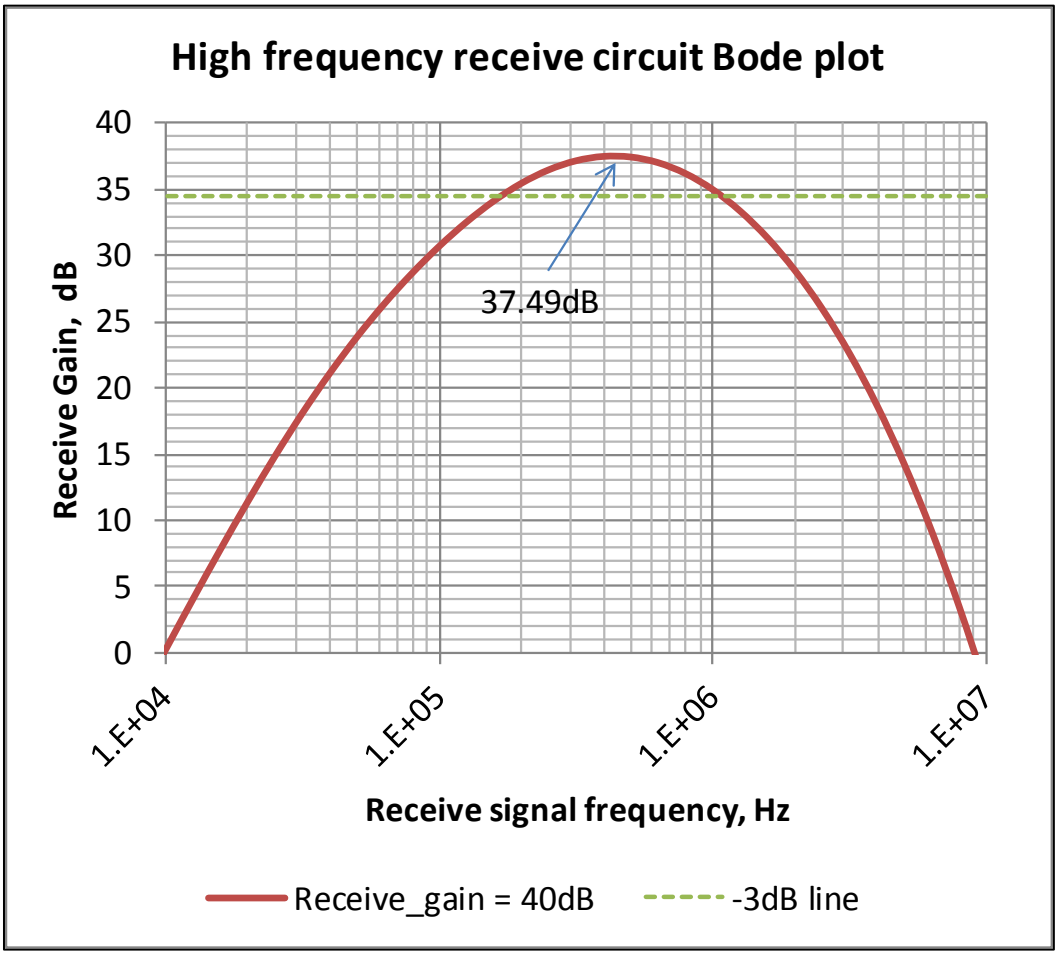


Figure 4-6 Bode magnitude plot, showing receive circuit's performance in simulation

4.4.4 Power supplies

The power supplies discussed in Chapter 3 were used with appropriate modifications, specifically to the capacitor bank (C_{bank+} and C_{bank-}) used in the high voltage power supply

discussed in section 3.4.1. In this application a 100 μF capacitor was used on the +Ve and –Ve rails of the high voltage power supply to provide a ripple-free supply to the transmit circuit. Simulation work was carried out using the developed simulation model of the high voltage power supply, but not presented in this section as it is similar to the previous work discussed in Chapter 3; section 3.4.1. However, the usage of the developed simulation model for verifying the design suitability for the application was demonstrated. In this work the high voltage power supply HTPSU was required to produce ± 100V. The ± 24V power supplies were modified to provide ± 15V DC supply for the input stages of the transmit circuit.

4.4.5 Typical system level simulation

This section outlines an investigation of the possibility of integrating the test specimen in the system model so that the receive signal strength at the receive ports of the pulser-receiver for a known transmitted signal strength can be observed in simulation. Modelling the medium and integrating in the simulation enables a study and consequently a definition of the port dynamics of the pulser-receiver unit. Accuracy depends on the accuracy of the model of the test specimen. The medium in most cases is a metallic component, for which the acoustic-mechanical property and the coupling between them are difficult to model in the electrical domain but very easily can be modelled in the EDA tools like Abacus (Avolution 2008) – a popular modelling tool for modelling acoustic and mechanical properties of materials. This research work was restricted to electrical simulation tool LTSpice and effort had been made to model the test specimen based on the work carried out by (Scott & Philip 2008) for wirelessly powering in situ structural health monitoring systems embedded within aircraft and other high value engineering assets. He used lossy transmission line models to model the electro-mechanical behaviour of the system.

The developed simulation models for a LRUT-transducer, transmit circuit and receive circuit were integrated with a simulation model for the medium (aluminium plate). This simulation demonstrates the ability of the system model in simultaneously simulating multi-engineering domain constructs for validation. The aluminium plate was modelled using a lossy transmission line model. The features of the aluminium plate were not included. Parameters used in this modelling are included in appendix A.

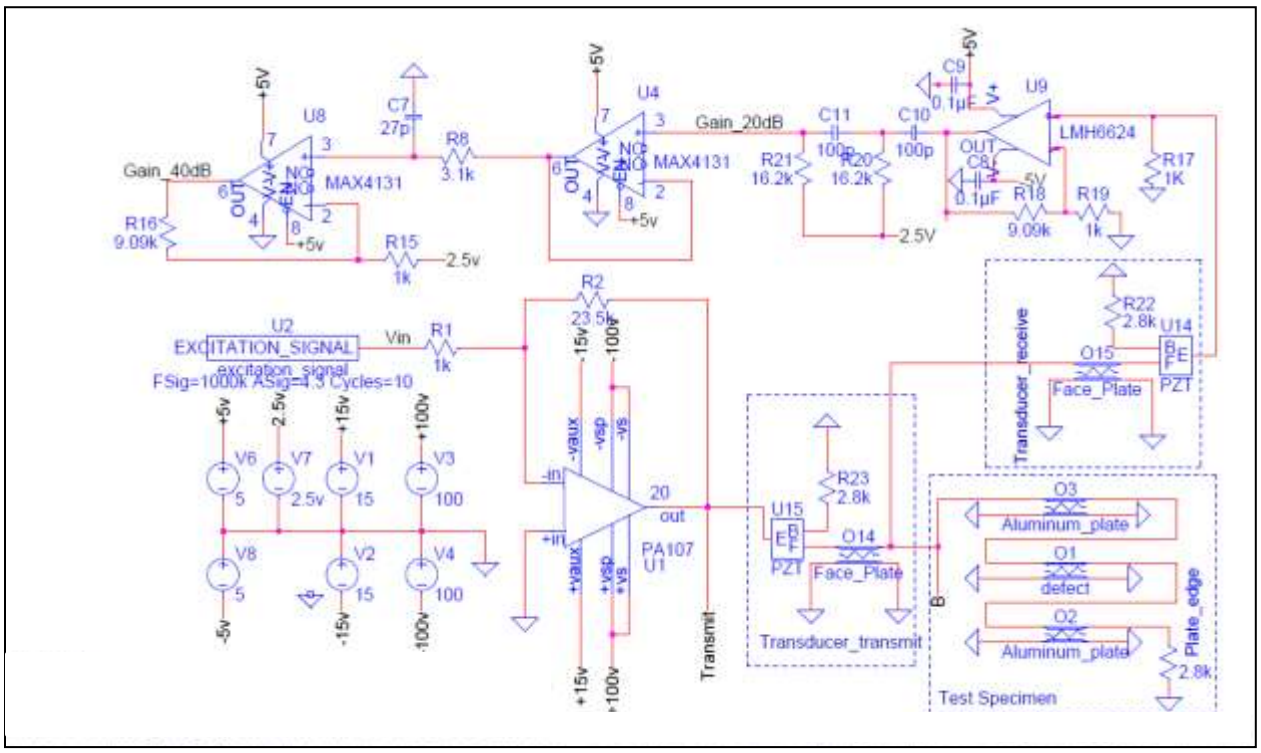


Figure 4-7 Multi-engineering domain system simulation of the medium frequency circuits

The high voltage power supply, HTPSU was not included in this simulation for the purpose of minimising simulation time and computing resource usage. This setup was used to simulate pitch-catch LRUT, using piezoelectric transducer models U15 and U14 as a transmitter and receiver respectively. The transmitted signal propagates through the test specimen Q3-Q1-

Q2 from the transmitting end to the receive end. Transducers U15 and U14 are assembled with face plates Q14 and Q15 and backing blocks R23 and R22 respectively.

The simulation results presented in Figure 4-8; a and b, show the receive signal strength, when a single transducer – U15 was excited with a ± 100 V; 1 MHz and ±100V; 500 kHz excitation signal respectively.

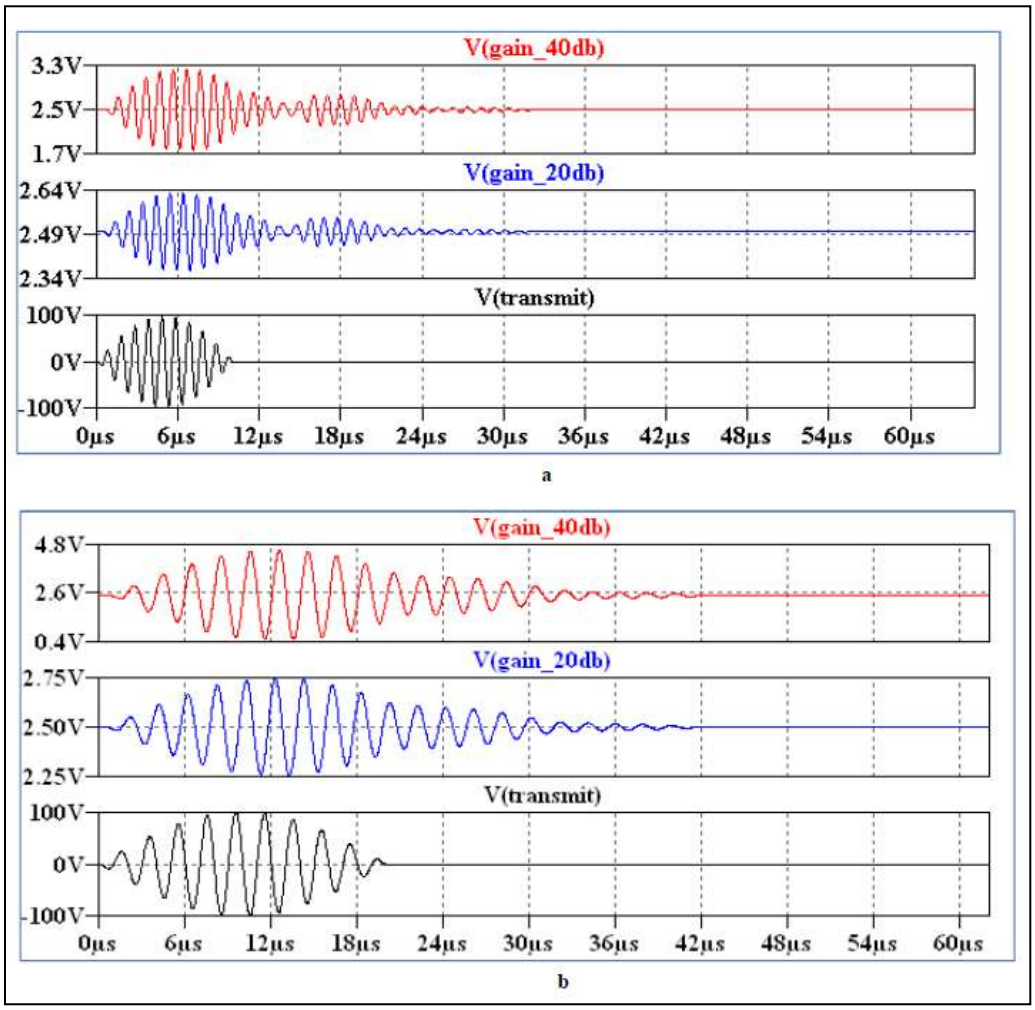


Figure 4-8 Medium frequency system simulation for excitation frequencies a: 1 MHz and b: 500 kHz

The signal received by the receive LRUT-transducer model (U14) was amplified by 20 dB at U9 and further amplified to 40 dB at U8 (40 dB in total). Wave mode separation is better

when an excitation frequency of 1 MHz was utilised compared to 500 kHz. This is noticeable in Figure 4-8; a, as the receive signal and the echo are well separated compared to their corresponding waveforms presented in Figure 4-8; b for 500 kHz excitation. However, 500 kHz excitation had better penetration, showing a higher receive signal amplitude in Figure 4-8; b, compared to the 1 MHz excitation presented in Figure 4-8; a. The models used to represent the medium are only approximate and cannot be used to predict application specific frequencies that (Haig & Stavrou 2012) predicted by FEA modelling of the test sample using simulation tool Abacus.

4.5 Hardware implementation and system integration

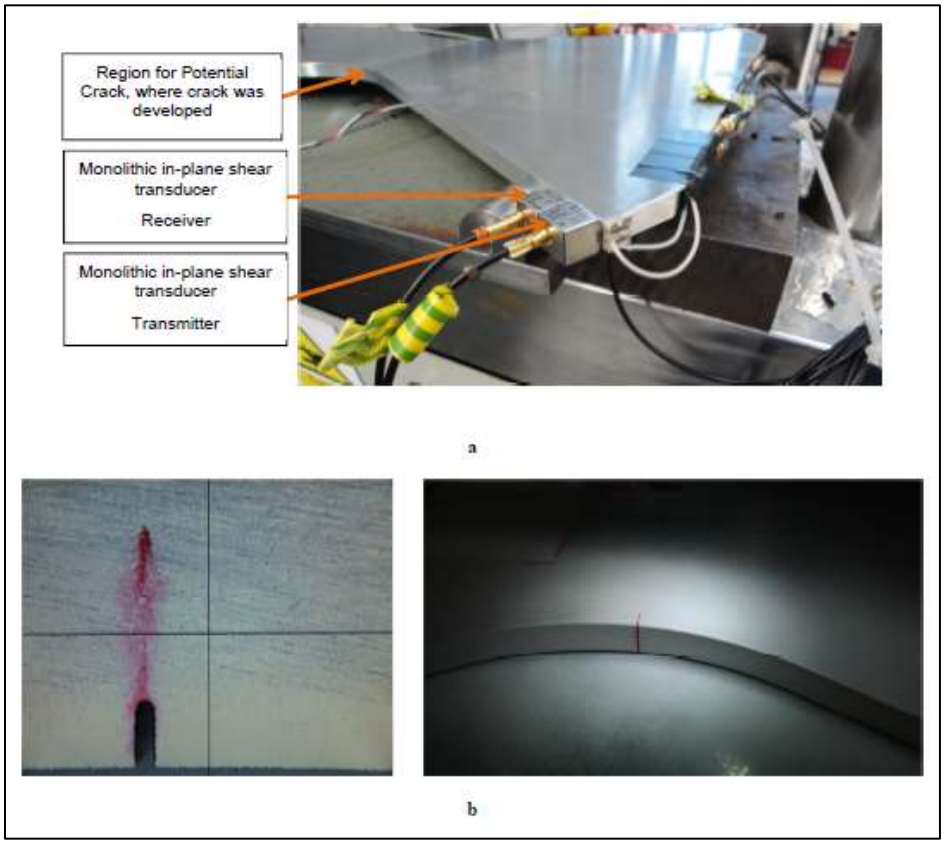


Figure 4-9 Experimental setup for high frequency LRUT on sample while developing a crack

An experiment shown in Figure 4-9; a, was setup as discussed by (Haig & Stavrou 2012). The prototyped hardware was used to condition monitor the test specimen using a medium frequency LRUT technique in pitch-catch mode, while a crack was grown/ developed from 0 mm to 5 mm in a controlled manner. The crack developed from a notch feature as shown in Figure 4-9; b.

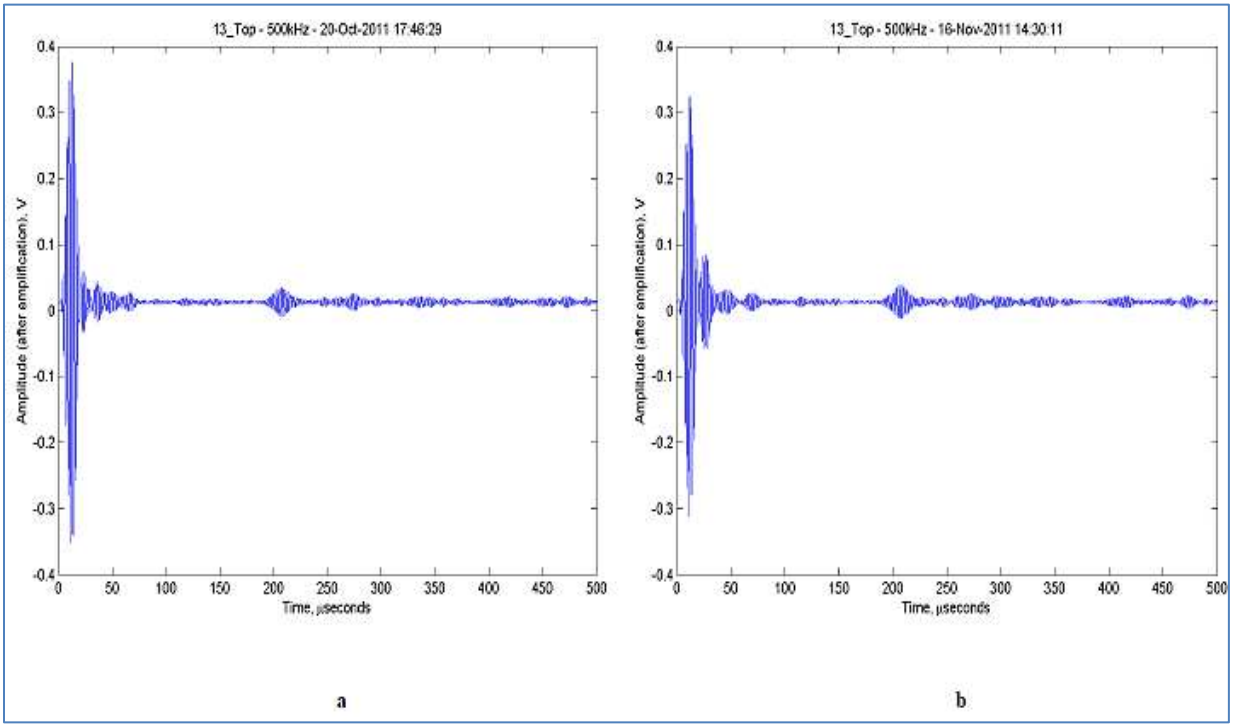


Figure 4-10 LRUT excitation at 500 kHz with crack size 0 mm (a) and 5 mm (b)

Medium frequency LRUT inspection results on the test specimen, when the crack size was 0 mm and 5 mm are given in Figure 4-10; a and b, respectively for an inspection frequency of 500 kHz. The plots show the received amplitude (after amplification of 20 dB) on the Y-axis and the time of arrival of the received signal from transmit time on the X-axis. It is noticeable that the received amplitude is approximately 380 mV when the crack size is 0 mm. Comparison of this practical results with the simulation results shown in Figure 4-8;b (250 mV) for 500 kHz

is outside the acceptable accuracy when it comes to wave-mode propagation analysis but is acceptable for the purpose of hardware development, as the order of magnitude is the same. This is important in defining the port dynamics of the hardware.

The developed hardware is at present implemented in a test rig that is subjected to a wide range of test/atmospheric scenarios for gathering large amounts of data for neural-network analysis.

4.6 Summary

This Chapter discusses the development of prototype hardware for a medium frequency LRUT application which was investigated by means of system modelling techniques. A medium frequency LRUT application was studied as a possible method of inspecting components with complex features. This application deployed pitch-catch mode, initially by means of exciting a piezoelectric transducer in shear mode, and consequently transmitting high frequency ultrasound in the 150 kHz and 1 MHz range through the test specimen of the material, aluminium. The ultrasound is received by a second piezoelectric transducer at a different location on the same test specimen. The LTSpice models developed in the previous Chapter were reused with necessary modification to develop this system and simulated for specifying and evaluating hardware. The model in particular included lossy transmission line models for representing a test specimen and transducer face plates, which in general possess acoustic-mechanical properties. This in effect dampens the ultrasonic wave generated by the excited transducers and resulted in the receive signal strength being reduced. However, these models are not optimised for analysing wave propagation analysis at present due to the dimensions and material properties used being

only approximate. Measurement data on the receive strength were observed to be in the same magnitude order as the simulation data.

The modular concept implemented in the system model development in Chapter 3 allowed relevant circuits to be modified and or enhanced without compromising the performance of the other circuits. The developed system model provided a platform for rapid prototyping besides the fact that the previously developed model was developed for power budgeting analysis in the first place. More work was required on the hardware for multiplexing the channels and optimising it for exploitation purposes, but this was not covered in this particular research work.

As a concluding remark, a powerful tool set has been developed in this part of the research work for future circuit enhancement, modifications and maintenance.

5 ELECTRON BEAM WELDING SYSTEM

5.1 Overview

This Chapter gives an overview of Electron Beam Welding (EBW) and describes the EBW system and its operation. An application specific problem, caused by a fault condition called flashover and its consequences is introduced, followed by a discussion on improving the welding process, so that weld quality and the integrity of the EBW system is maintained.

5.2 Electron beam welding process

Electrons can easily be extracted from a thermionic emitter (cathode), and accelerated by a high voltage, to convert the resulting kinetic energy involved into heat at the point of impact on the work specimen (Fritz et al. 1998). This is the basis of Electron Beam Welding. Conventionally electron beams are generated in vacuum ($< 5 \times 10^{-5}$mbar). Vacuum provides an insulation to avoid high voltage breakdown and protects the cathode (e.g. tungsten at 2700K) in the EBW gun from oxidation or erosion. It also prevents the beam spreading due to electron collision with gas molecules. Later researches showed that EBW at a reduced pressure (10^{-1} to 100 mbar) is also possible and has distinct advantages such as better control of weld defects (Sanderson & Ribton 1998), (Punshon, Sanderson & Belloni 1998). Non-vacuum EBW has also been recorded in the literature (Schulze & Powers 1998), (Fritz et al. 1998) although the inherent scattering of electrons through collision with molecules limits NVEBW to thinner sections of material.

The process is very popular in industrial applications such as aerospace (Russell 1981), marine (Galsworthy & Bird 1998), nuclear waste burial (Nightinggale et al. 1998), automotive

(Schulze & Powers 1998) and science for its high integrity, low distortion, higher welding speed, lower heat input, and greater depth-to-width aspect ratios than many of the other fusion type welding means (Schulze & Powers 1998), (Russell 1981), (Fritz et al. 1998). Although EBW and Light Amplified by Stimulated Emission of Radiation (LASER) beam welding have been widely accepted as high energy welding tools, EBW is superior to LASER in many aspects such as penetration level and energy capacity, but has the disadvantage of requiring the component to be within a vacuum chamber, which can reduce productivity, and requires biological shielding to absorb the X-rays produced by the electrons being stopped. Comparison of these two welding processes is very well documented in (Hanson 1986).

5.2.1 Electron beam Generation

Electron beam generation for material processing requires an EBW gun with a high voltage power supply and auxiliary power supplies. The EBW gun is an electrical device that releases a high energy electron beam when its thermionic cathode is heated and an acceleration potential is applied between the cathode and anode electrodes. The cathode is conventionally a tungsten ribbon filament heated using an auxiliary power supply, itself floated at the accelerating potential. However, the electron beam group at TWI currently uses an indirectly heated diode type gun for its more robust and long life cathode, simplified cabling and power supply requirements. Detailed description of the EBW gun and its technical details together with the performance are documented in (Sanderson & Ribton 1998). In essence, the main cathode (lanthanum hexaboride) in the gun is indirectly heated by a primary electron beam source based on a radio frequency (RF) excited (84 MHz) filament and accelerated under the influence of a constant negative 150 kV (-150 kV) electric field. The use of RF heating allows coupling of the

auxiliary supply for cathode heating across the gun vacuum, and precludes the need for auxiliary supplies floated at high voltage and the associated cabling.

The kinetic energy E produced by an electron when accelerated by an electric field V is given by Equation 5-1.The terms q, m and v are the charge of an electron (1.602E-19), mass of an electron (9.109e-31 kg) and the velocity of an accelerated electron respectively. Parameter V_r is the relativistically-corrected accelerating voltage that compensates for the electrons gaining mass. It is given by equation $V_r = V + 0.977 \times 10^{-6} V^2$), where V is the accelerating voltage (Klemperer 1959). For an accelerating voltage of 150 kV, the value of the energy produce is 2.403e-14 J, and the final velocity the electrons reach is in the range of 229.7E6 m/s. The power of the beam is adjusted by varying the beam current by heating the cathode. The value of the beam power is the product of the beam current and the acceleration voltage (Meleka 1971).

$$E = qV_r = \frac{1}{2}mv^2 \qquad \textbf{Equation 5-1}$$

5.2.2 High power requirement

A power density (W_b) of over 500 to 600 kW/cm^2 is usually required to obtain weld bead deep penetration by an electron beam. The power density (W_b) achieved in the presence of an acceleration voltage (V_b) and the beam current I_{beam} (1A = 6.28E18 electrons/s) is given by Equation 5-2 (Arata & Tomie 1986). K_s is a constant. It shows that the energy density is more strongly dependent on the beam acceleration voltage than on the beam current. In consequence it can be expected that higher voltage equipment will produce typically higher intensity beams with a welding performance characterised by higher depth to width ratios.

$$W_b = K_s\left(\frac{V_b^{5.1}}{I_{beam}^{1.7}}\right)$$ **Equation 5-2**

However, in the literature (Hanson 1986) (Arata & Tomie 1986) (Sanderson & Ribton 1998) (Punshon, Sanderson & Belloni 1998) (Binard & Ducrot 1986) (Russell 1981) it is evident that virtually all practical EBW systems available have a beam acceleration voltage limited to 30 kV to 175 kV. The upper limit of 175 kV is a constraint by the single-stage accelerating method (single-stage EBW gun) installed in such systems, and by practical constraints of providing biological shielding for the harder X-rays emitted by the electrons as they are decelerated at the workpiece. The literature makes evident that a higher acceleration voltage will increase the risk of high voltage breakdown in a single stage EBW gun. Experimental research work published by (Arata & Tomie 1986), demonstrated the possibility of using 300 kV and 600 kV acceleration voltages in five-stage and thirteen-stage EBW guns. In multi-stage guns the potential across each pair of accelerating electrodes is lower than the potential across the typical single-stage EBW gun, which acts favourably to supress the breakdown.

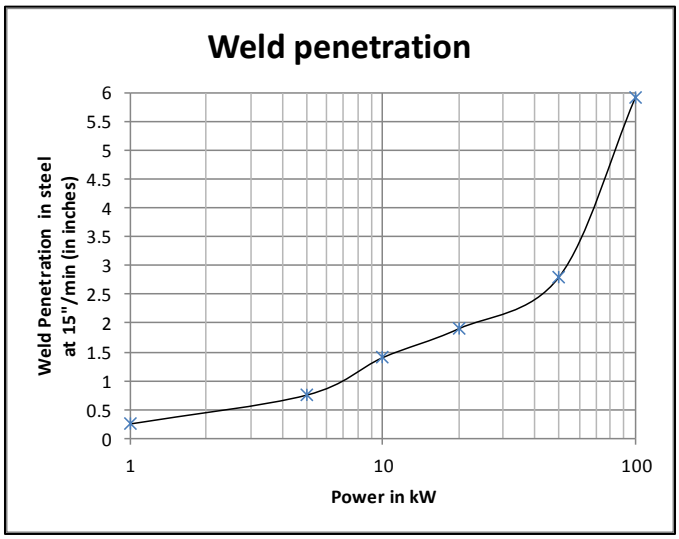


Figure 5-1 Weld penetration vs. electron beam power applied (reproduced from (Hanson 1986))

(Hanson 1986) in his work, presented practical results, claiming 100 kW power is required for fully penetrating a ~6” (~152 mm) thick steel test piece at a welding traverse speed of 15”minute^{-1} in vacuum. His results are reproduced in Figure 5-1. Work carried out by (Binard & Ducrot 1986) reveals a sample of thickness 300 mm requires a 200 kW gun for full penetration in a high vacuum. However, it is evident from later literature (Sanderson & Ribton 1998) that the weld penetration depth is strongly dependent on the beam intensity, although very narrow welds are generally not practical due to the high risk of joint misalignment, and the potential risk of weld porosity defects due to constrained liquid metal flow.

5.3 Description of the EBW System components

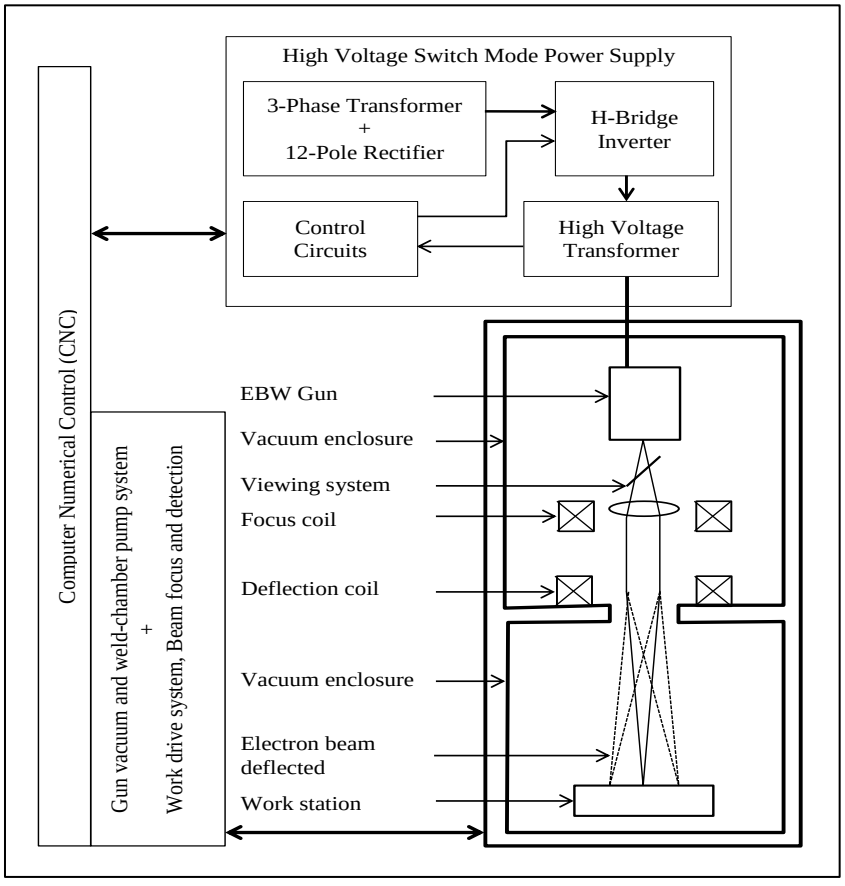


Figure 5-2 Simplified EBW system

A simplified diagram of an EBW system is shown in Figure 5-2. It shows the main components or subsystems that make the EBW possible. The physical construction of it is summarised in the following sub sections.

5.3.1 High voltage DC power supply

As discussed in section 5.2.2, this welding process requires a high voltage DC power source. Pulsed mode (Reinhold & Minkner 1965), and Continuous Mode (CW) (Reinhold & Gleyvod 1975), (Sanderson & Ribton 1998) power supplies have been built for Electron Beam Accelerators (EBA). Thick section metal fabrication (say 30 to 200 mm) generally requires full penetration welding or surface processing requires CW mode High Voltage; Low Current (HVLC) power supplies. From the literature survey, the topologies of power supplies that can achieve this are conventional 400 Hz motor-generator based power supply, Cascaded Cockcroft-Walton topology and inverter based Switch Mode Power Supplies (SMPS).

The EBW system at TWI is based on SMPS topology, to provide the acceleration voltage. It is composed of a diode converter (AC-DC), H-Bridge inverter (DC-AC), step-up transformer and diode rectifier. This supply is capable of providing 150 kV; 666 mA making it a 100 kW, CW power supply. It was favoured over the Cockcroft-Walton and conventional 400 Hz motor-generator power supply topology for the following reasons: For achieving high wattage at this level, a number of Cockcroft-Walton modules needs to be connected in parallel and each module requires individual controls and drives. At the time of development there were doubts on their reliability, particularly as this configuration requires many more components than a SMPS. In contrast the ability to produce power, exceeding the required level had been demonstrated by (Mizuno et al. 1989) and other TWI involved projects. Moreover, unlike conventional power supplies operating at a lower frequency, inverter based SMPS do not require large DC filters at

the output stage for ripple voltage reduction. The absence of the DC filters minimises the electric charge reservoir (stored energy) which works in favour during high voltage breakdown as the amount of charge present at the output is low once the inverter operation is terminated on over-current detection. The high frequency switching nature of the SMPS allows rapid control of the output which also provides a configuration that enables minimisation of the impact of flashovers.

5.3.2 Vacuum chamber

It has been universally recognised that EBW in vacuum achieves the unique and desirable EBW characteristics such as a high weld speed, lower heat input and greater depth-to-width ratios. Non-vacuum EBW can also be used to achieve acceptable performance for shallower welds with the help of higher voltage power supplies and specimen preparation (Schulze & Powers 1998). The vacuum level used for conventional in vacuum EBW can usually range from 10^{-2} to 10^{-5} mbar and the vacuum integrity in the EBW gun column and the weld-chamber, is considered as one of the most important aspect in the electron beam processing instrument. Detailed research on the vacuum and its effect on EBW are outside the scope of this research. However, understanding the discharge (flashover) formation in such low pressure gas regimes is important in the decision making of the fault recovery control sequence and power supply weld dead time if flashover related faults occur. Depending on the nature of the weld (vacuum, reduced pressure/non-vacuum) and size of the component the welding-chamber type (localised/ enclosed) can change.

5.3.3 CNC and auxiliaries

Computer Numerical Control (CNC) provides automation of EBW processes through control of the vacuum system, machine interlocks, the high voltage power supply and work

piece/gun manipulation. Detailed description of the CNC and the associated controls is not relevant to the scope of this research.

There are issues specific to the high voltage power supply such as ripple in the acceleration voltage and beam current, and high voltage breakdown. These issues cannot be avoided, and if not handled appropriately can degrade the weld quality and jeopardise the integrity of the power source. The ripple in the acceleration voltage is an aspect addressed at the design stage of the power source and the system inherits its ability to provide smooth output based on the design of the output stage. However high voltage breakdown is a scenario, referred to as flashover, which occurs due to gas or vapour ingress to the gun acceleration region from the weld-chamber and/or weld piece that degrades the insulation between the electrodes. This fault condition has the potential to trigger many more of the same or similar flashovers and can consequently damage the weld specimen and the power source.

The system is equipped with fault detection circuits, which detect a flashover and act upon it. This research focuses on the detection of this specific fault condition, the power supply control measures that can be taken to minimise its effect and to prevent propagation of a series of events and enhancing the system, so that the degradation of a weld and the safety of the power source is minimised.

5.3.4 Micro-discharge, flashover and prevention

Flashover is a set of phenomena that describes when the electrical insulation between the electrodes breaks down, sparks or arcs. It can occur when dielectric insulators become contaminated on their surface or become damaged. This type of event can only be resolved by maintenance of these parts or their replacement. However, during the course of EBW vacuum

contamination can occur causing the accelerating gap to breakdown. Left unchecked these breakdowns lead to roughening of the electrodes and the gun housing surfaces thus leading to an increased likelihood of subsequent events. This phenomenon in the EBW gun, results in interruption to the welding process typically leading to defects which are unacceptable in the product application. One such poor quality weld on a valuable 120 mm thick steel plate caused by major flashover is shown in Figure 5-3.

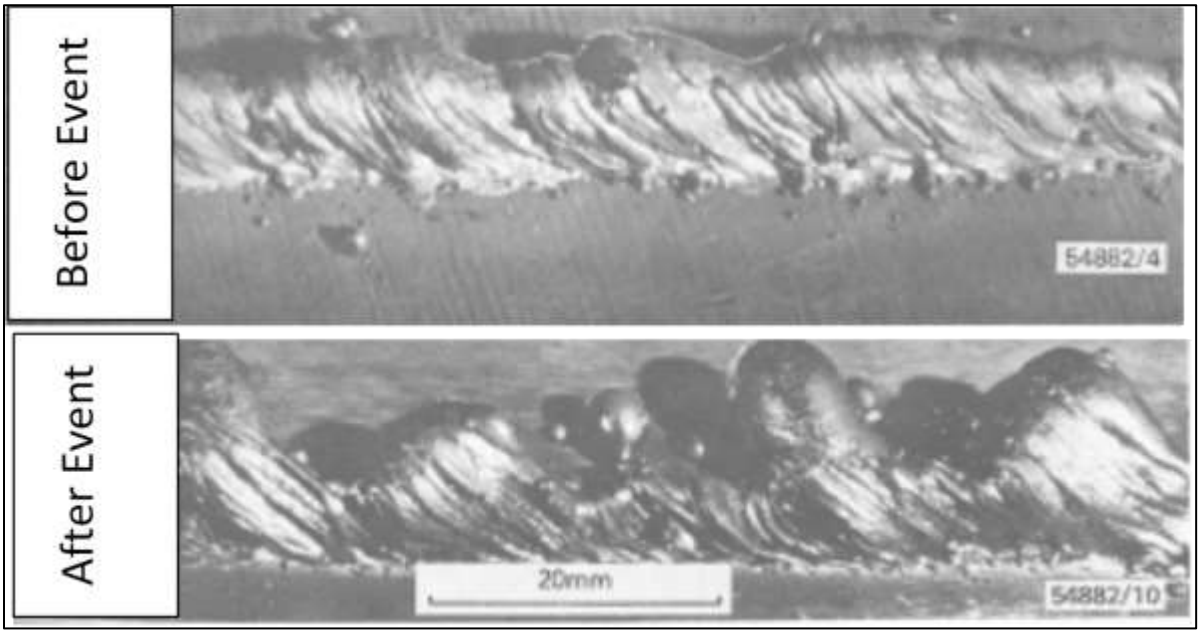


Figure 5-3 Weld defect caused by major discharge (Sanderson 1986)

Micro-discharge occurrence is probably due to an ion cascade mechanism occurring between system electrodes with the ion production taking place in the contaminant layers on the electrode surfaces formed by deposits from processing, pumping oil or similar. These self-extinguishing discharges draw only a small current and do not significantly disturb the welding process and begin to decrease some 20-500 μs after initiation (Peter & P 1976), (Sanderson & Ribton 1998). An operational procedure called gun conditioning is commonly employed where a high voltage is increased above the normal working level and the corona discharge is used to

outgas and decontaminate the gun (Sanderson & Ribton 1998). This process can erode microscopic features on the surface which otherwise would later make the gun prone to severe discharges (flashover). Flashover in effect pulls the acceleration voltage down to ground and interrupts the beam generation and weld process. It may also damage the power source and lead to high voltage-ampere characteristics, ending in the vacuum arc.

Metals that are welded in the EB weld chamber in normal conditions contain large amounts of dissolved gas and as soon as the electron beam impinges the weld piece gas is released with deleterious effect on the vacuum. This gas may scatter the electron beam or cause chemical contamination of the weld metal. Usually the EBW gun vacuum region and the weld chamber vacuum region are isolated and separate pumping systems are used to maintain the vacuum and in typical applications the EBW gun column is kept at a higher vacuum (10^{-5} mbar) than the weld chamber (10^{-2} mbar).

In practice prior to welding the gun is cleaned to prevent flashovers/micro-discharges. However this does not completely stop flashovers from occurring. There are mechanisms developed and implemented in the EBW systems in order to solve flashover problems such as a magnetic-trap system and improvements of the gun configuration (Arata 1986) (Sanderson 1978). These mechanisms are aimed at stopping/reducing the vapour generated in the weld pool getting into the gun column. However, the complete elimination of the risk of a flashover is not possible.

There is evidence that using a large capacity tube to absorb the surge energy of the discharge and re-establish the acceleration voltage after a short interruption can reduce weld interruption defects to acceptable levels. However, the use of such tubes must be close to the gun

(to minimise the energy stored after the tube in the HV cable). Such systems are also complicated by the feed of auxiliary supplies through the tube as well, and such tubes are of a high cost (e.g. £40k) and require replacement after some 2 years of operation. Alternatively, a modern SMPS utilised in the EBW applications switches at high frequency and this has the potential to allow rapid shutdown of the output once a flashover is detected (Sanderson & Ribton 1998).

5.3.5 System recovery after flashover

Modern SMPS that provide acceleration voltages are implemented with a control and regulating system capable of detecting the initiation of the discharge, blocking the beam emission and subsequently resetting the principal beam parameters to their original values within a few milliseconds. The EBW system of this type is in use at TWI and at present can shutdown the acceleration voltage (beam shutdown) and resume it back to the originally set value in 30 ms. This duration is called weld dead time, whose length of time is a conservative measure, that ensures welding does not resume until the vacuum integrity (and therefore high voltage insulation capability) of the electrode gap is re-established, and is short enough before the weld pool solidifies, which can lead to interruption defects in the weld body.

The literature published in (Gallagher & Pearmain 1983) reports that flashover establishment is very fast but not instantaneous. This delay time can vary for different material dust/gas, vacuum pressure, density and electrode gap and on average it can be as long as 10^{-4} s to about 10^{-9} s as the voltage increases. As for the self-extinguishing of micro-discharges, this duration is in the hundreds of microseconds range. The discharge duration depends on a number of parameters including mean-free-path of the residual gas present, electrode gap, residual charge left in the charge reservoirs at the output stage of the power source and speed of vacuum pumps.

The exact time value for extinguishing a severe flashover (discharge time) either using active dumping of surge charges or self-extinguishing is not explicitly stated in the literature, but (Gallagher & Pearmain 1983) suggests it is in the hundreds of microseconds. Expert opinion from TWI also endorses this and it has been concluded that the total duration of flashover establishment and discharge is 1 – 2 milliseconds.

Control circuits implemented by (Cazes, Dard & Sayegh n.d.), employs a weld dead time of 5 ms for 20 - 100 μs flashover occurrence for an EBW system at 60 kV with a 500 mA beam current; allowing a maximum of 1 ms for resuming an acceleration voltage to 60 kV and current 500 mA from 0. This system is based on a conventional power supply with a tetrode regulator tube, operating as a switch that turns on and off the acceleration voltage to the weld gun. A study published by (Shcherbakov 2012), investigated the main factors influencing the time characteristics of the discharge current, states that the time of restoration of the electrical strength in the welding guns (single stage) is 400 – 600 μs and after a time longer than the restoration time (usually 1 – 2 ms), the protection cycle of the power source is regarded as completed and the operating regime of the power source can be started. His study was based on an IGBT based inverter power supply which operates in the 100 – 200 μs switching regime.

5.4 Summary on findings

Despite the number of subsystems and processes involved in the EBW application, the power source holds a critical role in ensuring acceptable welding performance. Moreover, though there is a possibility of minimising the risk of flashover occurrence, by maintaining the gun electrodes and ensuring there is a clean weld environment, an absolute guarantee of flashover free welding is impossible due to the generation of vapour at the work piece.

The control systems that maintain the integrity of the vacuum cannot respond quickly enough to recover the integrity of the vacuum when there is vapour or gas ingress. Hence the only other possible method of reducing the risk of weld defects and protecting equipment is by the control of the I-V characteristics of the power source. Published work on discharge patterns by (Gallagher & Pearmain 1983), states that once a discharge has been established it can only be maintained as long as the voltage is above the discharge-extinction-voltage. The EBW power source is implemented with a fault detection circuit that handles a flashover fault condition. When a flashover is detected this circuit terminates the acceleration voltage for 23 ms before ramping it back to its original setting over the next ~ 7 ms, totalling 30 ms weld dead time. This weld dead time requires reviewing for the optimisation of the power source fault recovery control circuits.

Review and enhancement of the circuits rely on the understanding of two important aspects; firstly, regarding the discharge-extinction-voltage, no knowledge of this voltage level is sighted in the literature, and it can be assumed it can change according to the nature of the gas/ impurity present and the vacuum level. Secondly, the time it takes to re-establish vacuum integrity is not exactly known, although indications are it is in the single digit milliseconds regime. Consequently, the time the high voltage is held at zero, a lengthy 23 ms seems excessive. However the 7 ms taken to ramp the high voltage to its original setting can be optimised within the constraint that the power source and its control circuits can adapt to this rapid change in demand.

Moreover the 30 ms weld dead time set in the current design is long compared to the literature published and assumptions made. This reinforces the need for reviewing the fault recovery control circuits of this system, in order to enhance the performance of the welding

process without compromising the safety of the system and degrading the weld quality. Needless to say that optimisation of the control circuits of the power source rely on the performance of the circuits and components implemented in the power source. Having a full understanding of the power source and its capability is crucial for reviewing the enhancement options. The following section concentrates on introducing the EBW power source.

5.5 Introduction to the EBW power source

Electron beam welding applications require high voltage welding power supplies for accelerating the electron beam in order to achieve high energy. A single stage, EBW-gun allows the usage of a maximum acceleration voltage of around 170 kV, with reasonable resilience to high voltage breakdown. The EBW power source of interest is already in service and was designed to deliver 666 mA for thick section welding applications. In simple terms it is a 100 kW DC welding power supply.

The Switch Mode Power Supply is based on an H-Bridge inverter topology, driving a step-up transformer/rectifier to providing 100 kW DC power at 170kV. The output power, controlled by the beam current, can be adjusted from 0 kW to 100 kW. The system is a merger of three fundamental technologies comprising power conversion stages, power semiconductor device technology and control electronics. These three technologies and their contribution to the system are discussed in the following subsections:

5.5.1 Power conversion stages

Power conversion stages included in the concerned system can be described with reference to Figure 5-4 and Figure 5-5. The system can be sectioned as per the power conversion stage and Figure 5-4 block shows this diagrammatically. As can be seen the system evolves

through four different power stages for achieving high power from the utility supply. The topologies made this possible at each power stage are presented in simplified schematic diagram of the complete system in Figure 5-5. Each stage is discussed below:

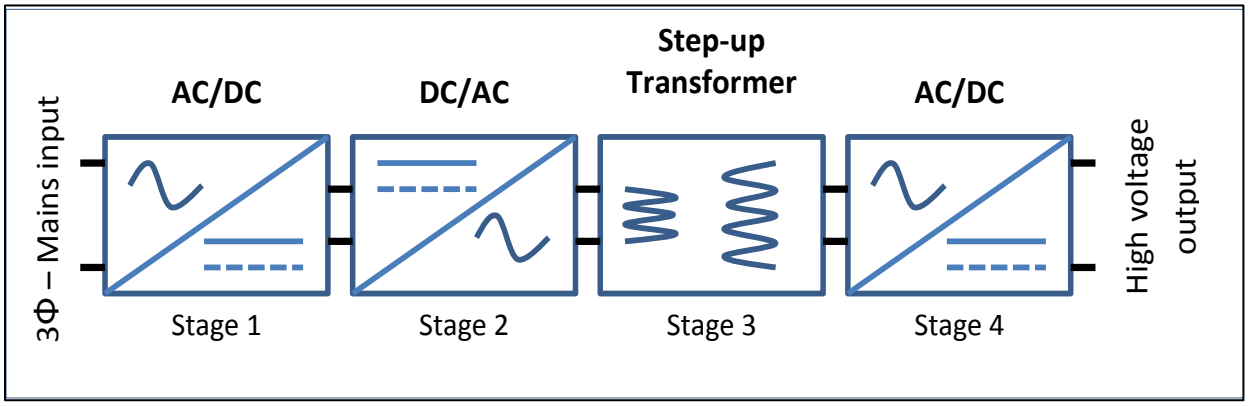


Figure 5-4 Block diagram of the high voltage power supply

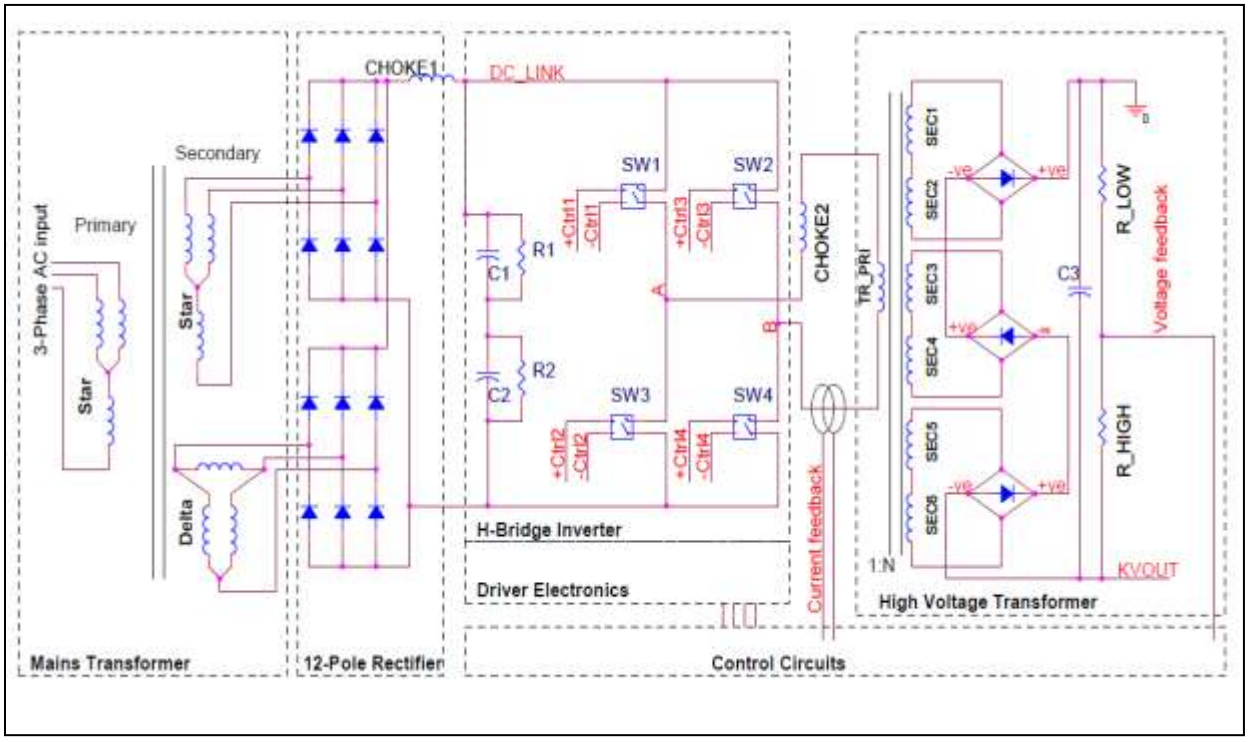


Figure 5-5 Simplified schematic diagram of the SMPS used in the EBW system

- **Mains transformer and 12 – Pole rectifier**

With reference to Figure 5-4, a utility supply provides the primary power to the power source. This 3-Phase 400V; 50 Hz supply is converted into a steady DC voltage at stage 1. This AC – DC power conversion is collectively performed by the electrical circuit labelled Mains transformer and the 12-pole rectifier in Figure 5-5. Its functionality is described as follows:

A star-star (Y-Y) and star-delta (Y-Δ) transformer arrangement is supplied with the utility supply. The Y-Δ transformer arrangement introduces a 30° phase shift between the primary and the secondary, whereas the Y-Y introduces zero phase shift. The topology is called a Y-Δ-Y phase-shifting transformer. It introduces a phase-doubler effect in the system, which is beneficial for reducing mains interference and harmonic distortion. The rectifier circuits – 12-Pole rectifier is an uncontrolled 12-Pole diode rectifier that performs the AC-DC conversion. It consists of two 6-Pole diode rectifiers that are supplied respectively by the Y-Y and Y-Δ transformers.

The introduction of the 12-Pole rectifier with the Y-Δ-Y phase-shifting transformer reduces the dominant 5th and the 7th harmonics that are seen in the line current of a common 3-Phase 6-Pole rectifier bridge based system. The unavoidable 11th and the 13th harmonics in the input current require suppression to prevent EMI spreading to the rest of the system and the mains. The rectified output DC voltage is notionally 600V although it can be as high as 622 V ($440 \times \sqrt{2}$) based on ± 10% tolerance on the 400V three phase supply in the UK.

The output of the rectifier is connected to stage 2, the H-Bridge inverter via a DC-Link choke (CHOKE1) for harmonic suppression.

- **H-Bridge Inverter**

The stable DC voltage produced by stage 1, is converted into an AC voltage waveform at power conversion stage 2. An H-Bridge inverter also referred to as full-bridge inverter converts the DC voltage at its input to the AC voltage that cycles at a sufficiently high frequency. The circuit labelled as the H-Bridge Inverter on Figure 5-5 shows the main components of its architecture which are briefly explained below:

The rectified voltage, labelled as DC-Link ($V_{DC\text{-}Link}$) is switched to the inverter's output terminal labelled as A and B in Figure 5-5, when switches labelled SW1, SW2, SW3 and SW4 are switched appropriately. These switches are turned on in pairs (SW1&SW4, SW2&SW3) for switching the inverter input voltage $V_{DC\text{-}LINK}$ to the inverter's output. The popular switching scheme - pulse width modulation (PWM) - is implemented for the control of the switches. The inverter output waveform across its output terminals A and B (V_{AB}), is a notionally square pulsed AC voltage waveform of amplitude $\pm V_{DC\text{-}LINK}$ or no pulses (0V) of frequency equal to the switching frequency of each pair of switches.

- **High voltage transformer and rectifier**

The inverter AC output, generated at power stage 2, is of a maximum voltage level of approximately 600 V. In this high voltage DC application, which requires a kilovolts voltage level, a voltage translation from the low voltage level of 600V to the kilovolts level is required. Power stages 3 and 4 collectively form the final power conversion stage of the AC – DC conversion. The topology realised in the hardware for achieving this is labelled as the high voltage transformer in the simplified schematic diagram shown in Figure 5-5. Its function in general can be explained as follows:

The inverter output V_{AB} is fed into the high voltage output transformer. As can be seen in Figure 5-5, a choke labelled as CHOKE2 is fitted in between inverter output V_{AB} and the high voltage transformer input. This choke and the transformer collectively form an inductive load across the inverter output V_{AB}. The reflected secondary impedance of the transformer to the primary side of the transformer, together with the parasitic resistance in the primary side, also influence in the characteristics of the inverter load. The high frequency nature of the V_{AB} permits reduction in the size of the magnetic core of this high voltage transformer compared with the lower frequency drives or mains frequency transformers.

Within the high voltage step-up transformer tank (oil filled for insulation) there are also high voltage rectifiers and a smoothing capacitor to provide rectification and smoothing of the high voltage AC voltage waveform produced by the high voltage transformer. The hardware also includes high voltage measurement with a voltage divider (R_High: R_Low) that provides a low voltage voltage-feedback signal to the control circuit. This voltage signal, referred to as the voltage feedback signal (V_{FB}) is utilised by the aforementioned PWM switching scheme for controlling the duty-cycle of switching.

The main transformer and the 12-pole rectifier circuit work together to provide the DC-Link voltage that is stored in a bulk capacitor reservoir of total value 17 mF (10 electrolytic capacitors of value 6800 μF/400V, arranged in a 5 parallel by 2 series arrangement with sharing resistors of value 7.5 kΩ across each capacitor). This is shown by the circuit arrangement formed by C1, C2, R1 and R2 in Figure 5-5. This bulk capacitor bank is aimed at reducing the RMS ripple current generated by the switching action of the H-Bridge inverter. Choke 1 is of value 0.5mH is a compromise between the level of 600 Hz current ripple and the physical size of the

choke. This DC-Link filter is anticipated to be much larger than that required for voltage and current ripple reduction.

5.5.2 Power semiconductor device technology

Power semiconductor switches are the most important devices in power converter applications. In this application special attention has been given to an industry leading power switching device Insulated Gate Bipolar Transistor, generally known as IGBT. These devices are utilised in the H-Bridge inverter. In contrast to other power switching devices such as power BJTs and power MOSFETs, IGBTs can operate switching with high voltages and are capable of carrying large current. Though detailed discussion on IGBT device physics is outside the scope of this thesis, its application specific semiconductor physics has been discussed briefly for clarity. This section also discusses the power diodes due to their importance in the success of inverter operation.

- Insulated Gate Bipolar Junction Transistor:

  IGBT, like BJT is a minority carrier device, meaning they have enhanced on-state performance. It is a three terminal device, whose terminals are referred to as collector, gate and emitter. The gate terminal possesses the physical property of the majority carrier device MOSFET, making these devices faster for commutation (on-off) applications. Because of these reasons, IGBTs are viewed as a device with MOS input characteristics and bipolar output characteristics. Hence it can be treated as a voltage controlled bipolar device, with high input impedance and large bipolar current-carrying capacity. Two traditional types exist, namely Non-Punch-Through (NPT) and Punch-Through (PT). PT IGBTs can be switched faster than NPT IGBTs, due to their extra N^+ buffer layer, introduced for reducing the tail

current at turn off. The introduction of a buffer layer however compromises the high breakdown voltage tolerance. Hence NPT IGBTs are preferred, in high voltage, medium frequency applications to PT IGBTs. This particular application uses NPT IGBT to handle voltages exceeding 1000 V. There is another type of IGBT, referred to as Trench-Stop for NPT IGBTs that has emerged in recent years. It has enhanced on saturation voltages and switching characteristics. These devices can also handle high voltages. The choice of NPT IGBT as switching devices in this application was based on their property of tolerating a high voltage and the availability at the time of development.

- Power diode

Power diodes, in this application are utilised in the uncontrolled 12-pole rectifier circuit and in the inverter application as an integral part of IGBTs (Co-Pack) as free wheel diodes. Power diodes in contrast to their low power counterparts have added complexity due to their ability to commutate several thousand amperes and accommodate high voltages. Their unidirectional current commutation ability, and fast reverse recovery property make these power diodes most suitable for high voltage uncontrolled-rectification. Characteristics of power diodes are very well documented in literature. This work concentrated on understanding the switching characteristics of the diodes due to the high *di/dt* experienced by the free wheel diodes in the inverter application. The rectifier operation of the 12-pole rectifier was not given attention because as will be discussed in section 5.7 this rectifier's performance in providing stable DC-Link voltage was considered satisfactory and its performance would not be affected by the planned optimisation process.

5.5.3 Control electronics

The performance and efficiency of the power converter rely immensely on optimised control electronics. There are control methods, such as traditional analogue PID control, microcontroller based PID controller, artificial intelligence control (fuzzy logic) utilised in power converter applications. Application Specific Integrated Circuit (ASIC) or FPGA based control electronics that incorporates analogue/digital conversion circuits are the most efficient way of controlling industrial power converters at present. The power source architecture concerned however, is implemented with the traditional discrete analogue PID controller based on the best and reliable topology available at the time of development.

The control mechanism, in simple terms, governs the switching function of the power converter switches and act on fault conditions. A PWM switching scheme is employed in this system for its ability to control force commuted switches. The switching scheme is implemented using a sufficiently high frequency with respect to the modulation function for minimising ripple in the load current and, and filtering for a ripple less load voltage. However, the limitations in power switching devices restrict the usable switching frequency. A moderate 5 kHz has been used with the aid of a popular PWM controller UC3825. The power supply is required to provide a stabilised high voltage level over the full power range and so the control is configured as a voltage stabilised system.

5.6 System enhancement constraints

The EBW power source is a complex architecture, making it difficult to carry out performance analysis and optimisation using real hardware. In addition to that, the large nature of the instrumentation makes access to test points difficult. The scale of the whole system is

symbolically shown in Figure 5-6. Attempting to force a flashover, in real hardware, puts the high voltage components at some risk. Hardware tests were also found to be difficult due to the restricted access to the system and availability of expert personal. Hence a system modelling approach was identified as an appropriate method of investigating this application sensitive issue of finding the optimum parameters for system performance enhancement. As previously mentioned, identifying the crucial components and functionalities that are sensitive to the specific analysis is an important step for the success of the modelling.

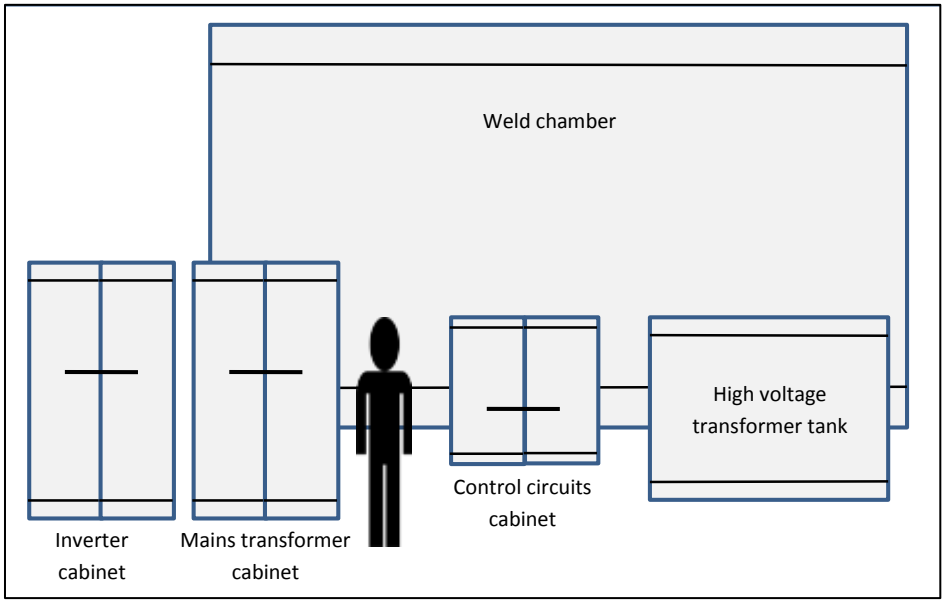


Figure 5-6 Large system makes access restriction

5.7 Identification of critical components

The introduction of a general concept of the EBW system, led to the important step of choosing the critical components that require investigation. Consistency and stability of a EBW rely on a stable DC-Link voltage. Practical work carried out showed a voltage ripple of less than 1% during high *di/dt* in the H-Bridge inverter. It is considered to be stable for the concerned system (Majed Dec., 1994). The steady delivery of the demanded welding power and the control

of it during and aftermath of a fault is highly dependent upon control of the H-Bridge inverter. The H-Bridge control is governed by the control circuits. The saturation effect and the impedance reflection of the high voltage transformer also influence the performance. The high voltage resistor network in the high voltage transformer fabrication possesses AC impedance characteristics due to the parasitic capacitance and inductance introduced during its component manufacture. This affects the response time of the voltage feedback signal for PWM control. Based on these facts the inclusion of the functionality of the following components/ subsystems in the system model was considered crucial:

- H-Bridge inverter and its associated components
- High voltage transformer and its saturation effects
- Control circuits
- Load behaviour

5.8 Conclusion

This Chapter introduced the concept of EBW and reviewed relevant literature on the application specific issues such as power generation and fault detection and fault recovery. Flashover is unavoidable in this high voltage, high power application and it has been identified that the current systems performance concerning fault recovery can be enhanced if the power source can withstand the stress of a rapid response. The complex and large nature of the system made performance testing using real hardware difficult. The research approached this problem by developing the crucial components and subsystems of the power source and simulating it in a PSpice environment. The following Chapter addresses the development of an EBW power source model.

6 MODELLING OF ELECTRON BEAM WELDING POWER SOURCE

6.1 Overview

This Chapter is dedicated to the modelling work carried out in the development of a PSpice based system model for an Electron Beam Welding power source. The developed model was aimed at replicating important functionalities of this power source and its control circuits in the form of a PSpice simulation model, so that a circuit level simulation can be carried out for functional analysis. This Chapter is a continuation of the previous Chapter, and focuses on introducing the crucial components and functionalities of the system in hardware terms and then address the methodology followed in order to construct the PSpice model that fits the purpose. Validation of the models is also discussed.

This Chapter, having demonstrated the modelling and validation of the crucial components and subsystems, introduces the top level system model of the power source, which comprises all the developed models of the components and subsystems followed by the system model validation. The Chapter concludes with two real application examples, where the developed model was simulated to review potential enhancement techniques.

6.2 Modelling of the H – Bridge inverter

The EBW power source is based on an H-Bridge inverter topology. It converts its DC input voltage (DC-Link voltage) into an AC waveform, by switching the DC-Link voltage at a specified switching frequency. The operation of an H-Bridge inverter is very well documented in

(Mohan, Robbins & Robbins 1995) and (Ali, Abdolhosein & Stoyan 2005) and is briefly explained in this section, with the aid of Figure 6-1 for completeness.

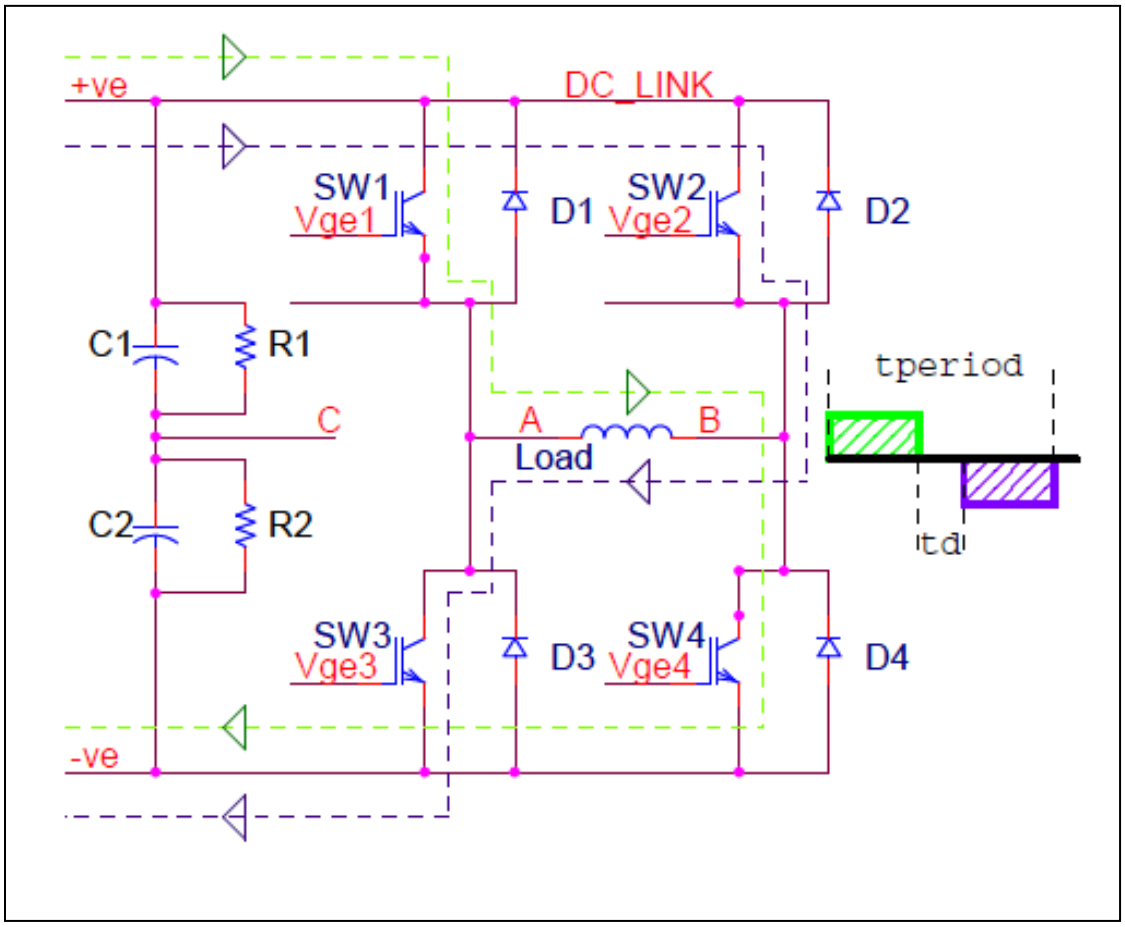


Figure 6-1 Simplified diagram of H-Bridge inverter

For simplification the circuit has been reduced to its main components. Its operation can be explained as follows: the four power switching elements SW1, SW2, SW3 and SW4 are semiconductor devices of type IGBT. They are operated in pairs. A load current is first allowed to flow through SW1, Load, and SW4 (green line) building a voltage potential V_{AB} ($+V_{AB}$) across the load (green pulse). Switches SW2 and SW3 are off during this commutation. After the pulse duration period, switches SW1 & SW4 are then turned off. After a short time period – dead time (td) the load current is allowed to flow through SW2, Load, and SW3 while SW1 and SW4

are off. This builds a negative V_{AB} ($-V_{AB}$) across the load (purple line). After the pulse duration period, switches SW2 & SW3 are then turned off. This process is repeated continuously to produce a train of square wave pulses, V_{AB}, the inverter output. During the dead time, the output V_{AB} is zero volts. A dead time is always present and it prevents shorting of the V_{DC_LINK}. Short circuiting the DC-Link voltage would lead to the destruction of the converter, due to an uncontrolled and extremely high current commutation through a vertical pair of switches. The maximum on-state time of each pair of switches in H-Bridge inverter applications is typically set to 80% to prevent simultaneous switching (Vinnikov & Laugis 2007). As will be seen later in this Chapter, the switching scheme implemented in this application, limits the maximum duty-cycle to 85% in its internal hardware. The switching frequency is set to 5 kHz by hardware.

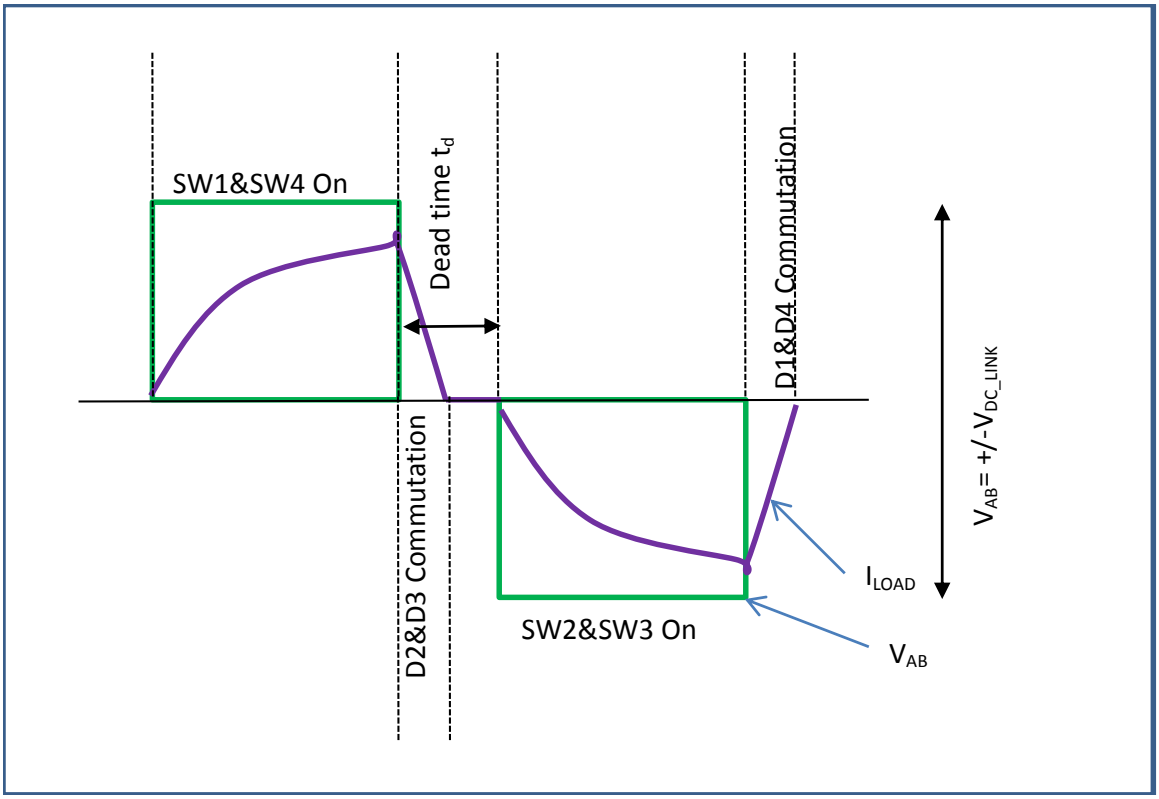


Figure 6-2 Diode commutation in inverter operation

Diodes D1 to D4 are called Free Wheel Diodes (FWD). These FWDs in effect make the unidirectional power switching devices SW1 - SW4 bidirectional to form a return path for the slow decaying load current. The inductive nature of the load introduces a discharge time constant that forces the load current to flow in the same direction, even after the switching devices are turned off. This time constant T_{LOAD} is the product of the load inductance and the parasitic resistance present in the load path. This is diagrammatically explained in Figure 6-2, highlighting the load current I_{LOAD} still flowing in the same direction even after the switches are turned off. These diodes also prevent voltage across the switching devices exceeding their breakdown voltages.

As will be explained later in this Chapter, the switching current in the inverter is expected to be in the region of 800 A. The inverter input voltage V_{DC_LINK} is as previously stated approximately 600 VDC. The IGBT devices of manufacturers part number DIM800DDM12-A000 from Dynex semiconductor are implemented in the inverter design as power switching devices for their high voltage blocking (1200 V) ability and high current carrying capacity (800 A continuous). DIM800DDM12-A000 is a Co-Pack device, meaning it is integrated with a FWD. Hence the need for external FWDs D1-D4 is avoided. Co-Pack packaging allows the inverters to be built with minimal stray inductance.

6.2.1 Effect of parasitic elements

Electro Magnetic Interference (EMI) is a major concern for inverter driven applications. The hard-switching of the switches and the nature of the high *di/dt* and *dv/dt* behaviour during the inverter operation cause EMI. The EMI can be reduced by appropriately constructing the inverter and using soft switching techniques, although this compromise leads to power dissipation in the switching devices. The dv/dt caused by the switch turn on (mainly) and off is

coupled through the parasitic capacitance between the switch substrate and the module base plate. In general, the base plate of the switching devices is grounded through the heat sink they are mounted on. Hence a noise current is generated and flows into ground and flows through the stray capacitance in the system. The high *di/dt* related EMI is mainly caused by the fast turning off of the switching devices and fast snap-off of FWDs. Inverter operation related EMI issues are covered in detail by (Tang 1998).

Stray-inductance (L_{STRAY}) around the switching devices is a parasitic inductance, unintentionally introduced during the mechanical assembly and device fabrication. This generates high voltage spikes, V_{SPIKE} at high *di/dt*. The value of this voltage can be evaluated using Equation 6-1. The parasitic capacitance, introduced by the stray capacitance (C_{STRAY}) across the switching devices causes high current spikes on the DC-Link bus bars. C_{STRAY} is generally generated by the stray capacitance between the collector and the base plate of the switching devices. The current spike, I_{SPIKE} can be calculated using Equation 6-2.

$$V_{SPIKE} = L_{STRAY} \frac{di}{dt} \quad \text{Equation 6-1}$$

$$I_{SPIKE} = C_{STRAY} \frac{dv}{dt} \quad \text{Equation 6-2}$$

Equation 6-1 and Equation 6-2 conclude a high EMI can be reduced by reducing L_{STRAY}, C_{STRAY}, *di/dt* and *dv/dt*. The optimisation of parasitic inductance and capacitance is generally done during the device fabrication and mechanical assembly of the inverter. The Co-Pack packaging of IGBT packages allows reduction in L_{STRAY} and C_{STRAY}. Once the mechanical assembly is carried out, the inverter inherits the parasitic parameters as its property. Hence, the possibility of enhancing the EMI performance can only be dealt with electrically, i.e. by controlling *di/dt* and *dv/dt*. In addition to FWDs, introducing snubber circuits and soft switching

techniques are the usual remedies that reduce EMI and protect switching devices from overvoltage and over current situations. Appropriate switching circuits also help. The Co-Pack allowed a snubber free design in this application.

Behaviour of the switching devices IGBTs and the FWDs in circuit operation vary from one design to another according to the load. In addition, the parasitic elements unintentionally introduced during fabrication also influence their performance. This application identified, using appropriate models for the IGBTs and the FWDs are important to simulate the inverter functionality.

6.2.2 Modelling of IGBT and FWD

Fundamentals on IGBT devices and diodes are very well documented in (Mohan, Robbins & Robbins 1995) and (Sattar 2001). The IGBT used in the TWI inverter has manufacturer's part number DIM800DDM12-A000 from Dynex semiconductors. This is a Co-Pack module with integrated FWD (Dynex semiconductor 2009). It is rated at a breakdown voltage - V_{CES} = 1200 V and continuous collector current - I_C = 800 A. A PSpice simulation model for this device is not available from the manufacturer and has not been sighted in the literature. A physical model for an IGBT device DIM800DDM17-A000, which is from the same family of DIM800DDM12-A000 has been developed by (Ramy et al. 2004) and been used intensively for device characterisation (Ramy et al. 2008) (Azar et al. 2004). This model is based on a Kraus model (Kraus, Turkes & Sigg 1998).

Despite the claimed fitness and accuracy of the DIM800DDM17-A000 physical model in device characterisation, this published model failed to converge in circuit level transient simulations with moderate simulation parameters. A similar physical model had been created for

DIM800DDM12-A000 using parameters extracted from the literature. This work is documented in (Parthipan Sept, 2011). However this model was found to be unfit for this research work for the following reasons:

- This work failed to reproduce the device characterisation results published for DIM800DDM12-A000 (Dynex semiconductor 2009). The efforts made to reproduce device characterisation data for DIM800DDM17-A000, using the published model (Azar et al. 2004) (Ramy et al. 2008) also failed to match literature (Dynex semiconductor 2009).
- The device specific physical parameters were not available for DIM800DDM12-A000. This restricted refining of the developed physical model for this device.
- The physical models developed by (Ramy et al. 2004) and (Parthipan Sept, 2011) for DIM800DDM17-A000 and DIM800DDM12-A000 respectively struggled to converge at circuit level simulation even with moderate simulation settings causing simulation failures and long duration computation.

The complexity of the EBW power source and the need for tight tolerance in circuit level simulation requires a simplified model with acceptable accuracy, so an alternative model for DIM800DDM12-A000 was required. The literature very well records a number of different approaches for modelling empirical, semi-mathematical, mathematical and semi-numerical IGBT models. A publication (Kuang, Barry & Stephen 2000) reviewed all IGBT models published in the literature. It analysed, compared and classified the models into different categories according to type, objectives, complexity and speed.

PSpice provides IGBT models based on a Hefner model (Hefner, Jr 1991), (Hefner, Jr 1995), (Orcad Inc. 1999) in the PSpice model library. A PSpice model editor template allows inputting device specific parameters that can be extracted from manufacture's datasheets. Hefner documented a parameter extraction procedure for his model. This can be found in (Joakim 2002). This procedure helps produce device specific parameters that are in general unpublished by IGBT manufacturers. However, these involve building test circuits using IGBT devices and carrying out practical measurements. The unavailability of a standalone DIM800DDM12-A000 device and lack of resources to build required test jigs prevented executing this parameter extraction exercise. In this research a device model of CM1000HA-24H (Mitsubishi Electric 1998) provided in the PSpice library had been chosen as a bench mark and its model parameters were adjusted as per the DIM800DDM12-A000 datasheet. Simulation results were obtained by simulating the device model for device characterisation and comparison with the manufacturer's datasheet. This process was repeated until a close fit was achieved.

The Hefner model provided by PSpice does not include FWD, so a diode model was modelled using a PIN diode model template provided in the PSpice. Power diode parameters were extracted and adjusted from the published model for DIM800DDM17-A000. The developed models for the IGBT DIM800DDM12 – A000 and its internal FWD are included in appendix B.

6.2.3 IGBT and FWD model evaluation

The IGBT model was characterised in simulation to produce and refine its output and switching characteristics to match the corresponding parameters published in the datasheets of DIM800DDM12-A000 (Dynex semiconductor 2009). Similarly, the FWD model developed was also characterised using simulation to match the forward and reverse characteristics of the real

component. Characterisation of these models for these parameters was crucial to tune the model to replicate the real switching performance of the inverter and simulation correctness at a system level. The outcome of this work and comparison discussion is included in this section.

- IGBT Output characteristics

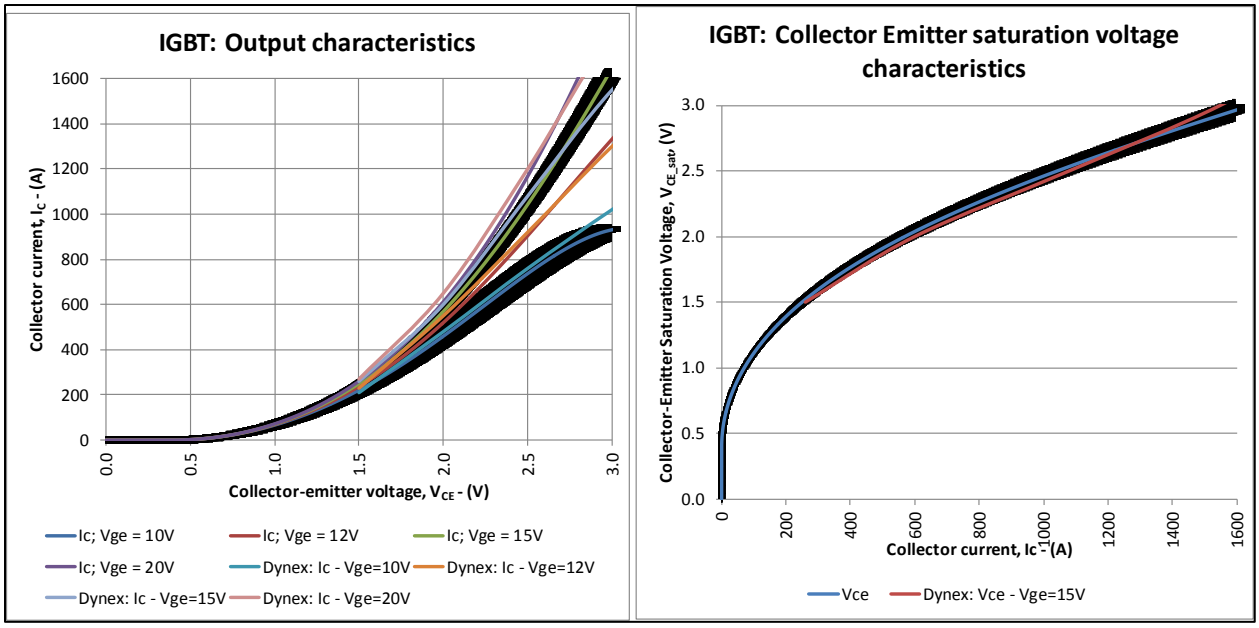


Figure 6-3 IGBT output characteristics comparison

The output characteristics define the value of voltage across the collector and emitter terminals of an IGBT (V_{CE}), when it conducts a given collector current, I_C for a given value of the gate voltage across its gate and emitter terminals (V_{GE}). The IGBT is intended for a switching operation only and the range for its practical use is limited to the range of V_{CE} within the saturation area.

The graph presented in Figure 6-3; left, shows the simulation results obtained by simulating the developed PSpice model for the Dynex IGBT device, DIM800DDM12 – A000. It also includes manufacturer's data for comparison with error bars set to ± 1% offset at all data

points on two specific simulation datasets V_{GE} = 10V and V_{GE} = 15V. Error bars on other lines are not set for clarity. Manufacturer's data was extracted from the datasheet of DIM800DDM12–A000 (Dynex semiconductor 2009) by a curve-fitting method and reproduced. This method only used five distinctive points on the characterisation graph published in the datasheet; hence the reproduced graphs are more linear in contrast to the published curved graph. This was considered acceptable as the in-between interpolated points would only be approximate values if more points were sampled. Moreover the reproduced graphs are informative enough to be used for device selection or comparison. Measurement data on the characterisations were not available due to limited resources. The device's saturation curve for the gate-emitter voltage V_{GE} = 15 V, is presented in Figure 6-3; right was also produced, using the same data (manufacturer: dynex; I_C - V_{GE} = 15V, simulation: I_C – V_{GE} = 15V). This graph in contrast to Figure 6-3; left, plots the data in I_C vs. V_{CE_SAT} format for clear interpretation of device saturation voltages. Like its counterpart, this plot is also presented with error bars set to ± 1% offset at every data point of the simulation dataset. The literature (Dynex semiconductor 2009) does not include the graphical interpretation of the saturation effect; however it states the saturation value for this device as 2.8V maximum and 2.2V typical, for the test conditions V_{GE} = 5 V; I_C = 800 A @ 25° C. Simulation results also reveal the IGBT model has a saturation voltage of 2.25 V for a collector current (I_C) conduction of 800 A. Simulations were carried out with the simulation temperature set to 27°C. The saturation voltage characteristics graph published for a similar device CM1000HA-24H from a different manufacture Mitsubishi (Mitsubishi Electric 1998), has a similar trend to Figure 6-3; right, not showing the extent of the saturation voltage for increasing the current above rated value. This is because above twice the specified I_C value, the device is assumed degraded.

Deviation between the manufacturer's data and the simulation results is below ± 1%. The justification for setting the error bars to ± 1% is as follows: There are three important option settings for a PSpice simulation for analogue simulations, namely RELTOL (relative accuracy of V's and I's), VNTOL (best accuracy of voltages), and ABSTOL (best accuracy of currents). These parameters by default are set to 0.001, 1 μV and 1 pA for RELTOL, VNTOL and ABSTOL respectively. This means that a node with a voltage 1 V is accurate to 1 mV and small values such as 10 μV are accurate to only 1 μV. Similarly, a current 1 A is accurate to 1mA and the accuracy can go down to 1pA. However, 1 pA accuracy is meaningless for IGBT operations as they are voltage controlled devices and their internal I_{DS} (drain-source current of gate construct) is very much higher than 1pA, exceeding 100s of μA. For transient simulations of power electronics, RELTOL = 0.01, VNTOL = 1 mV and ABSTOL = 1 mA is adequate and having these settings help in achieving convergence (Shilpa & Taranjit 2003). Based on this, a saturation voltage value of 2.2 V (typical) can be revealed as 2.2 V ± 22 mV. This is similar to ± 1% of 2.2 V (1% of 2.2 is 0.022) as the error bars are set. I_C is the parameterised factor; hence the error bars are discussed for V_{CE} only. RELTOL of value 0.01 will make the smallest time step in the simulation to (simulation-time)/1E-13 s.

In conclusion, the output characteristics from the simulation results closely map the manufacturer's data.

- IGBT switching characteristics

The switching characteristics of the IGBTs are very much affected by the stray inductance and capacitance of the inverter build. The gate drive circuit also influences the operation. Hence an IGBT's, switching behaviour can vary from one system to other. However,

modelling and simulating the switching nature of the IGBT on its own is equally important, as the IGBTs on their own inherit parasitic capacitances during their fabrication that affects their switching performance. Figure 6-4; a, depicts the parasitic input capacitance C_{ies} (C_{ge} + C_{gc}, with output short circuited) and the output capacitance C_{oes} (C_{ce} + C_{gc}) associated with a typical IGBT. Note the turn-on time and turn-off time of IGBTs are not absolute values, and they change with collector current (I_C), case temperature (T_C) and the value of the gate resistor (R_G). However values for these are published in datasheets, for specific values of I_C, T_C, and R_G at constant V_{CE}.

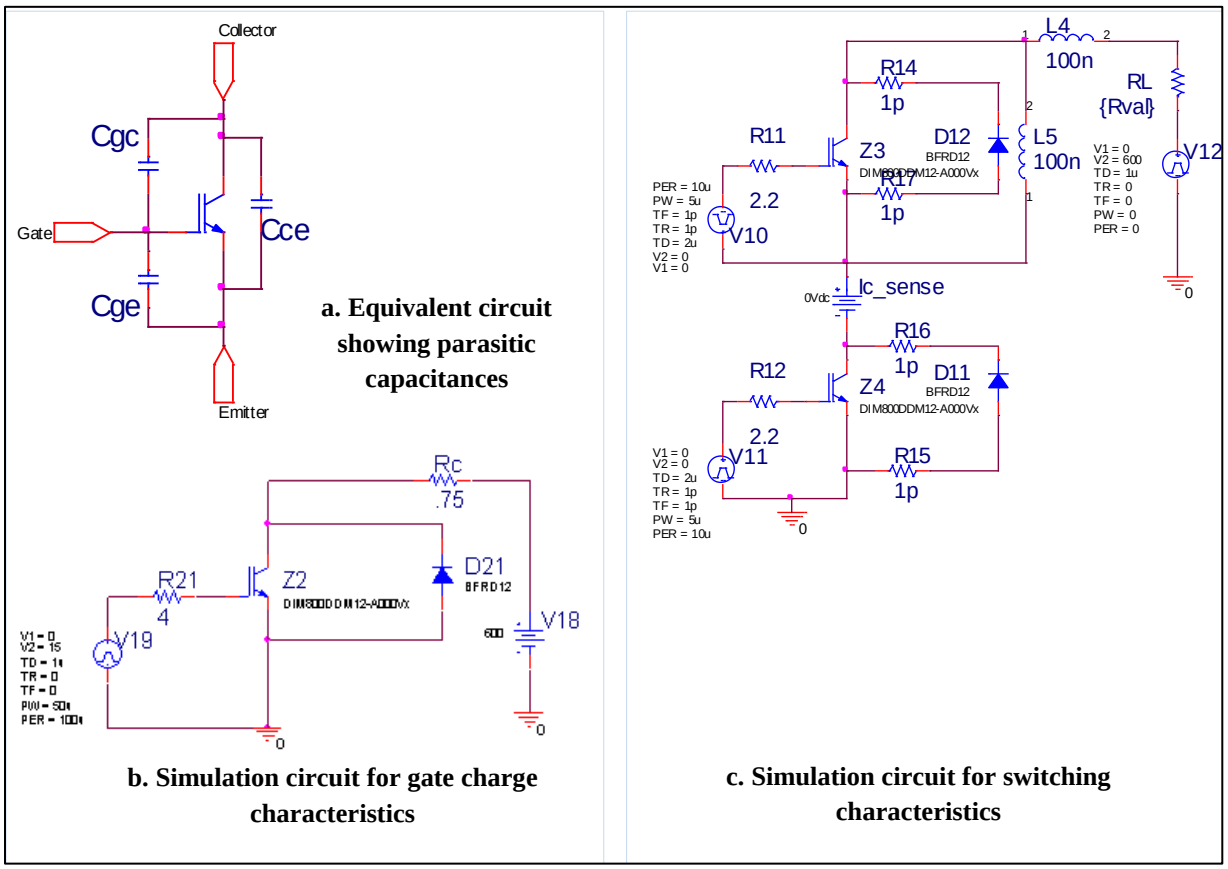


Figure 6-4 Simulation circuits for gate charge transient and switching characteristics (right)

The IGBT model developed for DIM800DDM12 – A000 was simulated for analysing the gate charge characteristics of the device. The simulation circuit Figure 6-4; b permits analysing the turn-on aspects of the device model. The simulation results obtained are presented in Figure

6-5; left. The same circuit can be used for analysing turn-off aspects of the device as well. The turn-on aspects of the developed model for the DIM800DDM12–A000 are described as follows:

The turn on of the IGBT is a current phenomenon and is explained with the aid of Figure 6-4; a, b, and Figure 6-5; left. At turning on a transient, a positive gate voltage V_{GE}, causes the gate current to charge the input capacitance of the IGBT C_{ies}. Once the V_{GE} reaches the IGBT threshold voltage V_{TH} (at end of time t_1), the collector current I_C starts to flow. By this time the C_{ge} is fully charged. The gate-emitter voltage, V_{GE} continues to increase, charging the C_{ies} and at the end of time t_3, full conduction is accomplished, with I_C reaching its final value, and V_{CE} at saturation voltage. The decrease in voltage V_{CE}, causes an increase in $C_{gc,}$ which is why there is a slow increase in V_{GE}, between time t_2 and t_3. Simulation results presented in Figure 6-5; left, depicting the aforementioned turn-on process, reveals the threshold value (V_{TH}) for the IGBT model as 5.6V. Similarly the turn off process can also be performed, but not included in this section as the fitness of the model in terms of the turn-off parameters is demonstrated using a different approach – the switching time simulations.

Figure 6-4; c, shows a half bridge inverter simulation circuit that allows the switching time of the device model to be studied for inductive loads. With reference to Figure 6-5; right, the parameters included in the legend are defined in Table 6-1. The turn-on time t_{on} ($t_{d\ (on)}$ + t_r), and the turn-off time t_{off} ($t_{d\ (off)}$ + t_f) are important when drive circuits are considered. It is also worth pointing out that the gate capacitances vary with different V_{CE} and hence the t_{on} and t_{off}. The rise time is basically limited by the gate impedance characteristics, which are partially a function of the gate contact geometry and partially a collective time constant formed by R_G and C_{ies}. The turn-off delay time, $t_{d(off)}$ is due to gate capacitance. The fall time t_f is the tail period,

introduced by the recombination of excess charges in the gate's n-region. It is not influenced by the device capacitance. Analysis on IGBT switching is very well documented in (Powerex Inc. n.d.), discussing all aspects of the IGBT switching at a semiconductor level and for component level analysis, it is in general, the gate resistor R_G influences the turn-on and turn-off time.

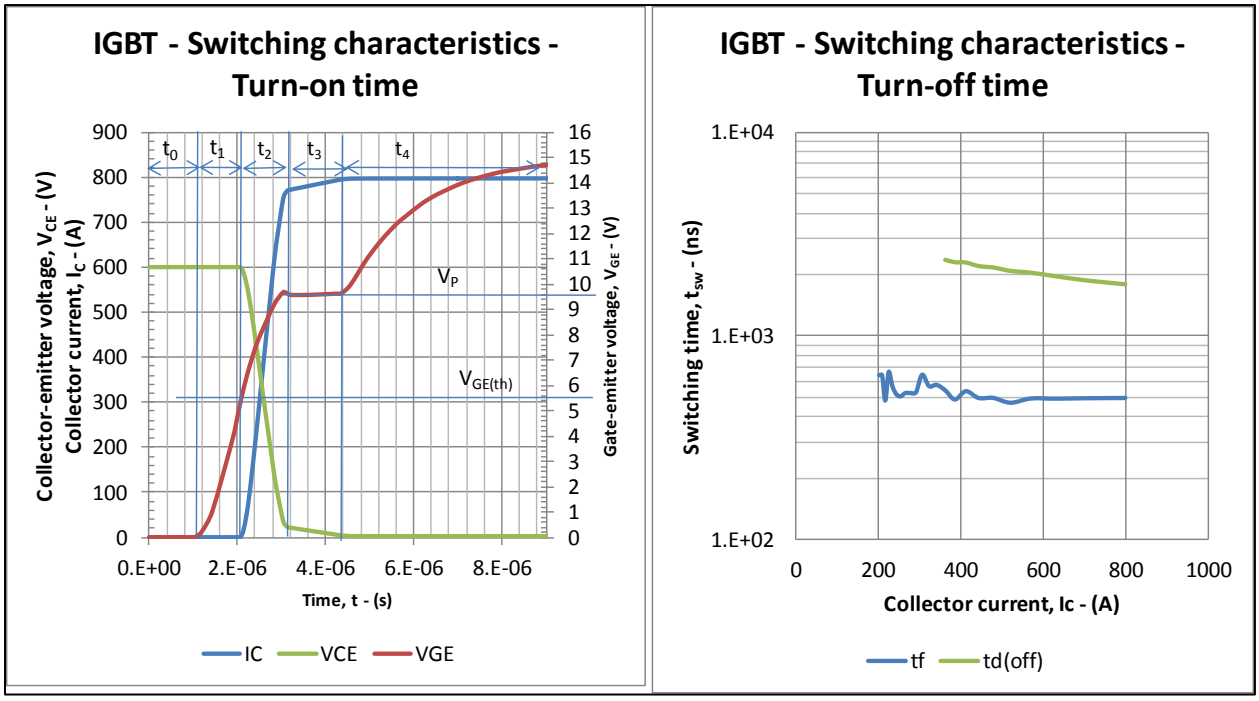


Figure 6-5 Simulation results showing switching characteristics

Parameter	Symbol	Description
Turn-on delay time	$t_{d(on)}$	the time required to attract excess electrons to the region just underneath the gate
Turn on rise time	t_r	the time for I_C to rise from 10% to 90% of its final value
Turn off delay time	$t_{d(off)}$	delay time due to gate capacitance limiting charges from leaving the under gate area
Fall time	t_f	the time for I_C to drop from 90% to 10% of its final value

Table 6-1 IGBT switching transient Device parameters defined (Mitsubishi Electric 1998)

Figure 6-5; b, presents the simulation results, obtained for the turn-off fall time t_f and the turn-off delay time $t_{d\ (off)}$ for increasing I_C. For I_C = 800 A, values for t_f and $t_{d\ (off)}$ are approximately 500 ns and 1800 ns respectively. The data sheet published figures for t_f and $t_{d\ (off)}$

as 170 ns and 1250 ns for the same conditions. The accuracy of these results was compromised by the PSpice simulation transient options setting; RELTOL = 0.01 and the simulation time TSTOP = 500 us. In general, the time resolution in a PSpice simulations is limited to $(RELTOL \times TSTOP)/10$), giving the time resolution of 500 ns (Microsim Corporation 1997).

Like other power switching device models, IGBT modelling has a trade-off between accuracy and simplicity. However, the simulation results for device characterisation showed acceptable accuracy.

- Power diode characterisation

The power circuit simulation requires the modelling of the transient behaviour of the FWDs integrated within the IGBT COPAK DIM800DDM12 – A000. The PSpice model editor provides a template for creating diode models. This template is based on a Lauritzen model that allows the diode's forward characteristics, and reverse characteristics to be modelled (Ma & Lauritzen 1991), (Chen et al. 2011). This is a physical model and requires some device specific physical data that are not available in this case. The model also requires some other parameters that can easily be extracted from manufacturer's datasheet.

The IGBT manufacturer Dynex semiconductor provided device specific physical data for one of their power diodes used in the higher voltage IGBT Co-Pack, DIM800DDM17 – A000. This power diode's performance is similar to the one integrated in the DIM800DDM12 – A000, the IGBT COPAK device used in this application, except the breakdown voltage is higher for the diode model, provided by the manufacture. The model was edited appropriately and used in this simulation work.

As the FWD is an integral part of the COPAK, the manufacturer's datasheet does not provide a great deal of information on its performance. However, its typical forward voltage characteristics are published in the datasheet, together with values for its reverse recovery current (I_{rrm} = 380A), diode reverse recovery charge (Q_{rr} = 80 µC) at test condition: diode forward current (I_F = 800A), V_{CE} = 600 V, and $\frac{dI_F}{dt} = 4200\ A/\mu s$. Note, that these values are not an absolute property of the device, meaning these values can vary depending on the test condition. The value for the reverse recovery time t_{rr} is not given.

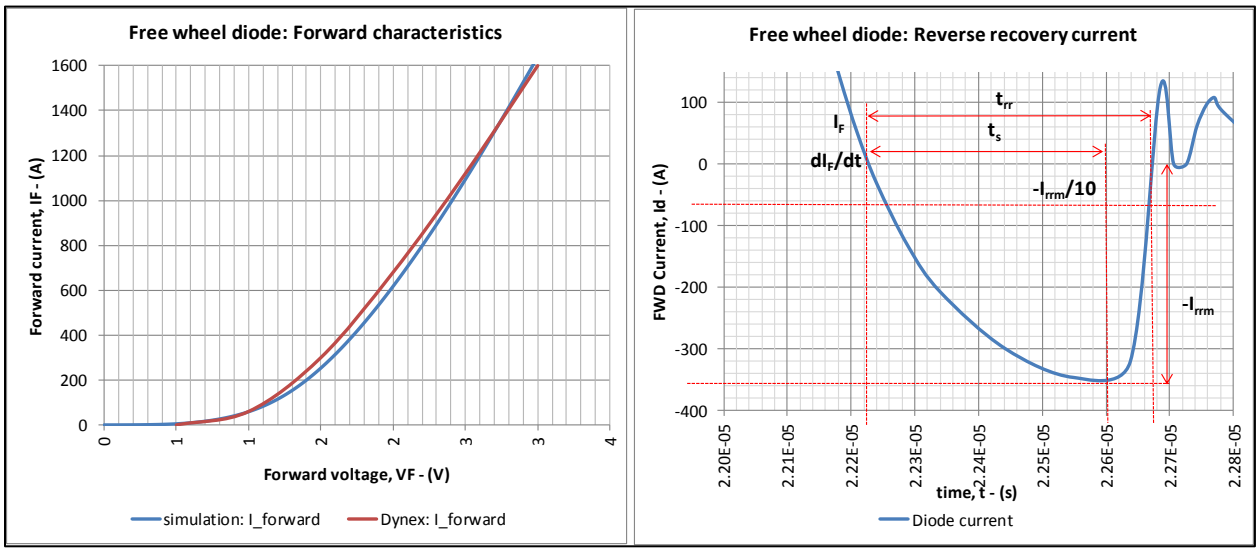


Figure 6-6 Free wheel diode forward and reverse recovery current characteristics

The transient behaviour of the developed diode model was simulated. The simulation results obtained are presented with the manufacturer's data in Figure 6-6; left, for the forward characteristics of the FWD. The manufacturer's data were extracted from the datasheet, by a curve fitting method. Close matching was observed between the simulation results and the manufacturer's data.

Data for evaluating, the diode's reverse recovery behaviour was not available in graphical format. However, the simulation results were produced using the simulation circuit shown in Figure 6-4; c, and presented in Figure 6-6; right, for demonstrating the ability of the model. The test conditions set for this simulation were: diode forward current (I_F = 800 A), V_{CE} = 600 V, and $\frac{dI_F}{dt} = 4070\ A/\mu s$. The factor $\frac{dI_F}{dt}$ was adjusted by adjusting the R_G value (R11 = 1.2 Ω, R12 = 1.2 Ω), for achieving ~ 4000 A/μs to match the datasheet test condition. The simulation values for t_{rr}, and I_{rrm} were extracted as 450 ns and 360 A respectively. These values closely match the reverse recovery characteristics published for the FWD. The value for the diode reverse recovery charge Q_{rr} was derived as 81 μC using formula, $I_{rrm} \times t_{rr}/2$. The snap factor S is approximately 0.83, derived using formula,$S = (t_s/t_{rr}) = (350\ ns/450\ ns)$. The Snap value highlights the potential threat of EMI during turn off. Note that values for t_{rr}, I_{rrm} and Q_{rr} differ with the varying factor, dI_F/dt. Increasing this ratio leads to higher values for t_{rr}, I_{rrm} and Q_{rr}.

6.3 Control electronics modelling

The control electronics utilised in the system perform two functionalities; control of the system output power characteristics, and fault detection and recovery control. The system's output characteristics are controlled by varying the duty-cycle of the switching signals that govern the turn-on and turn-off of the power switching devices, IGBTs, in the inverter. The term duty-cycle is a ratio between the turn on time and the period of the switching cycle. A switching scheme, popularly known as Pulse Width Modulation (PWM) facilitates this. The circuit also provides overcurrent detection. When an overcurrent is detected the driving signal is masked out until the next driving pulse. This is handled by the off the shelf IGBT driver module (Power integrations 2000) on detection of the overcurrent signal generated by the control circuits. A

flashover response is triggered when the overcurrent occurs more than 10 successive half cycles. The demand signal is reduced to zero for 23 ms. This is a dead-time set to allow vacuum recovery. This period is followed by a ramp-up period of 7 ms for the demand signal to ramp-up to its original set value. This fault recovery period is known as the weld dead time. These circuits play a vital role in the control and stability of the EBW power source, and with appropriate modifications to these circuits, the output power characterisation and the systems response to faults can be enhanced. The following sub sections discuss these two aspects.

6.3.1 Output power characterisation and system stability

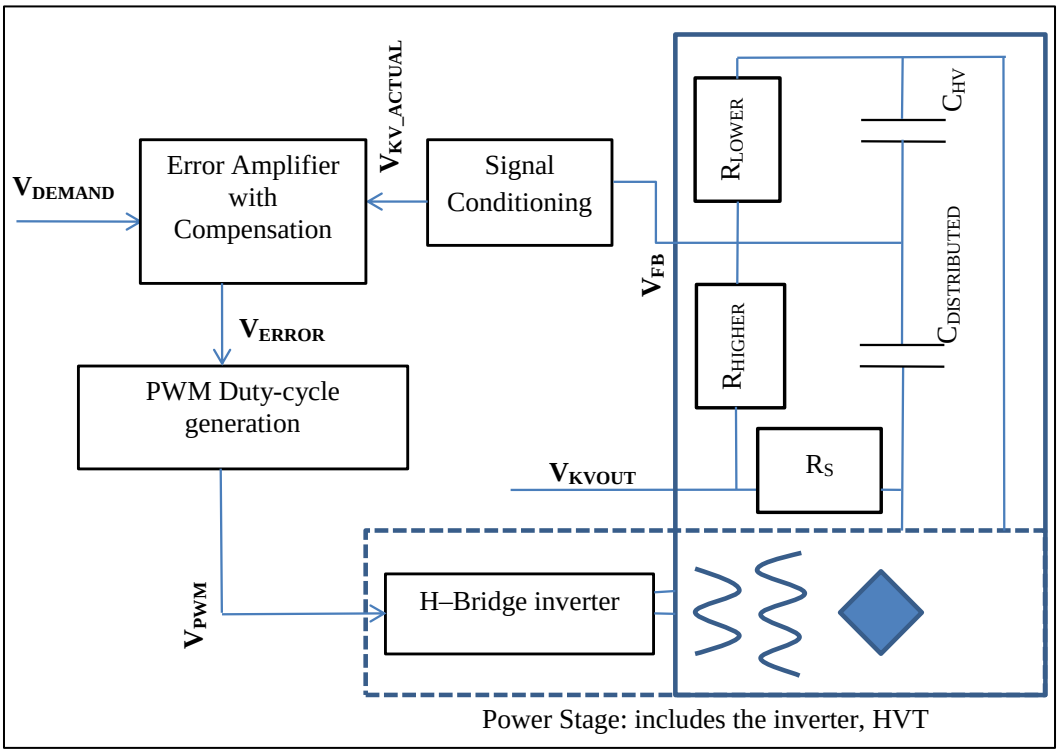


Figure 6-7 Voltage-mode feedback control of the SMPS for voltage output - V_{KVOUT} stability

The power source output characteristics can be modified by varying the inverter switching signal duty-cycle (Basso 2008). A feedback control, based on voltage-mode control, is implemented in the control circuits with the aid of a commercial PWM controller UC3825.

Voltage-mode control is preferred over current-mode operation because it is a voltage regulated power supply. Regarding the operation of this voltage-mode control mechanism, reference is made to Figure 6-7.

The error-signal V_{ERROR} is obtained from the difference between the demand-voltage V_{DEMAND} and a portion of the power supply output voltage - V_{KV_ACTUAL}. The voltage divider realised by R_{LOWER} and R_{HIGHER} produces the required portion of the power source output voltage – V_{KVOUT}. In real hardware, a ratio of 1:7500 is implemented with the input stage of the signal conditioning instrumentation connected parallel to R_{LOWER} (V_{KVOUT}/V_{FB} = 7500). Further attenuation of -12 dB was implemented in the signal conditioning stage for adapting a 300 kV = 5V scaling on V_{KV_ACTUAL} signal at the input of the error amplifier. A capacitor C_{HV} was included in the design to compensate for the distributed capacitance $C_{DISTRIBUTED}$, which was unintentionally introduced during the fabrication of the high voltage resistor network R_{HIGHER}. The resistor R_S serves the purpose of limiting the absolute minimum of the secondary impedance to its value (2 kΩ) to prevent a 0Ω short circuit at the output under fault discharge conditions.

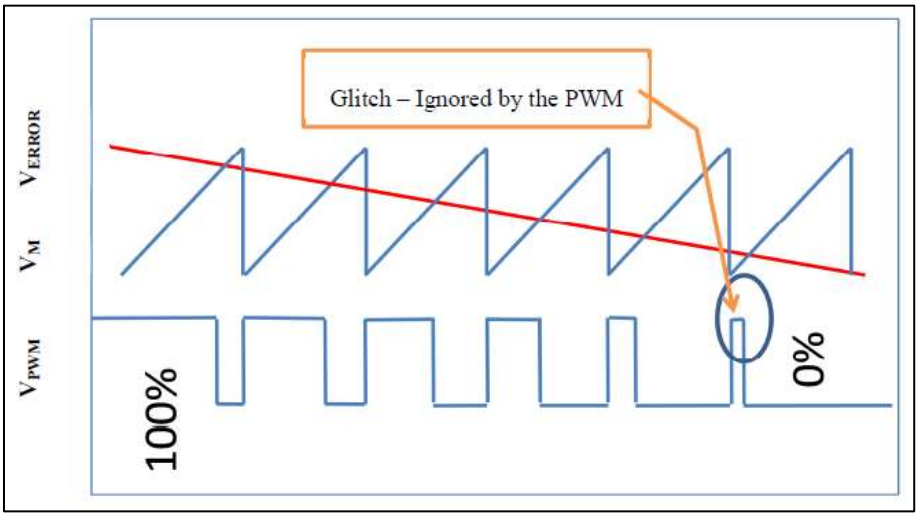


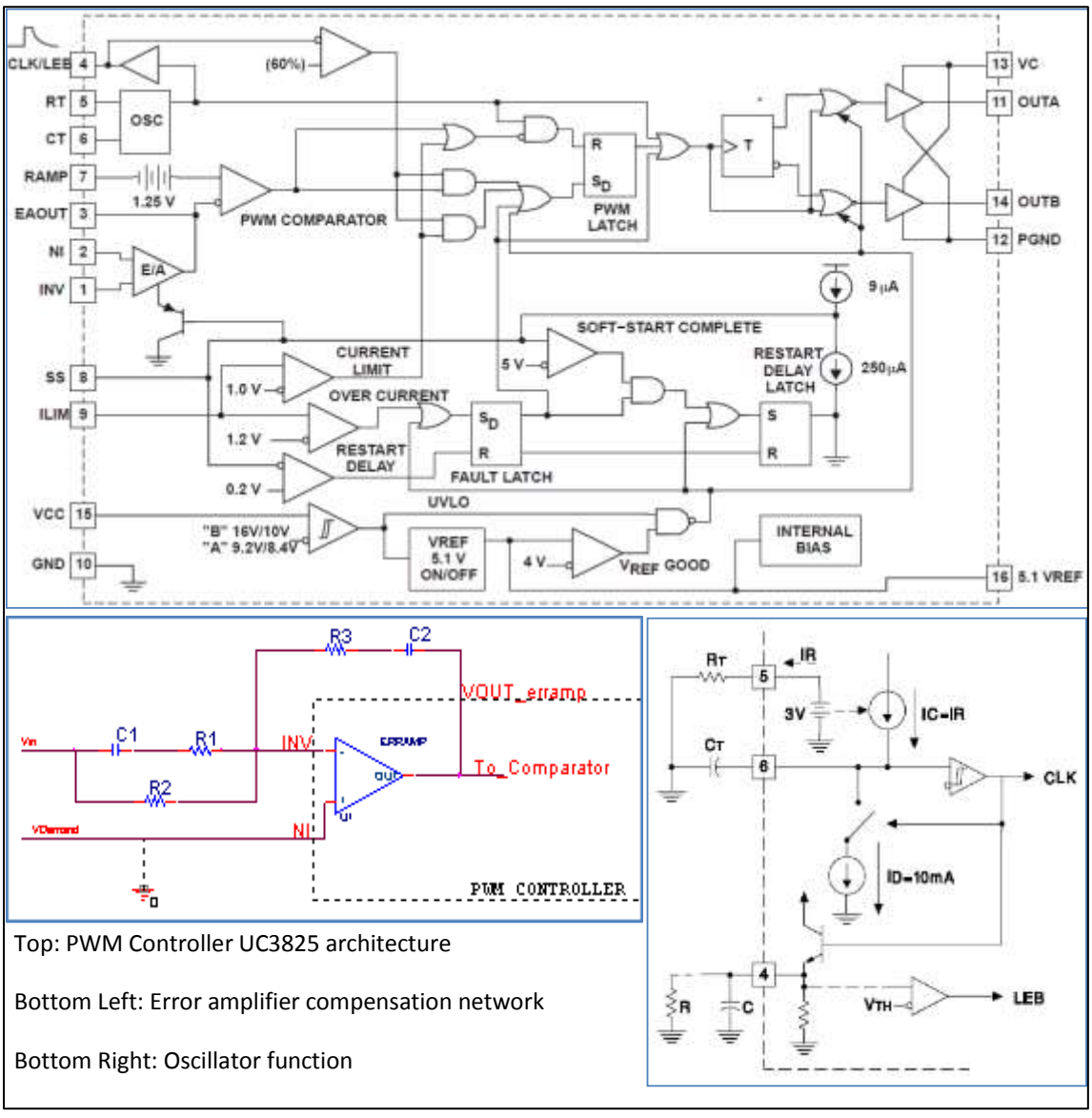
Figure 6-8 PWM signal generation

At the PWM stage the error signal V_{ERROR} is permanently compared to a saw-tooth waveform (V_M) of fixed frequency, approximately 10 kHz in this particular application to produce a transition on its voltage comparator's output signal V_{PWM}. This signal is then fed through appropriate drive-electronics to switch the IGBTs in the inverter. Figure 6-8 portrays the aforementioned operation. The duty cycle of 100 % down to 0% is possible in principle. However, UC3825 limits the duty-cycle to 0 – 85% for safe operation of the inverter.

The system stability hence relies heavily on the performance of the PWM operation, especially on the response of the error amplifier (ERRAMP) output to the change in input – V_{KV_ACTUAL}. The compensation network built around the ERRAMP allows the output response to be modified in a way to stabilise and output - V_{KVOUT}. The error amplifier circuit shown in Figure 6-9 is implemented in the real hardware to achieve stability. The original values chosen were by experience and then the circuit had been tuned by a trial and error method during the system commissioning stage. As predicting frequency response of the power stage is difficult, optimising the system stability was predominantly carried out in the time domain in this application. Optimisation of transient dependent systems for stability by means of a frequency domain method is inadequate because the frequency domain analysis linearises the nonlinear components models during simulation, making valuable information unavailable for analysis.

A PSpice vendor Cadence provides a PSpice macro model for a PWM controller SG1825, based on a SG1825 datasheet. This device's specification resembles the specification of a UC3825 controller that is used in the real hardware. Hence the SG1825 model has been used in this simulation work, though a state-space average model has been documented in (Shih et al. 1995) for UC3825. This model is based on functional description of the device UC3825 with minimised details on propagation delays, current and slew-rate. As a consequence of its

simplicity this model failed to include the output stage of the controller, which is relevant for more realistic transient simulation. This work was an extension of (Bello 1984) and based on the proposition published by (Middlebrook & Slobodan 1976) on state-space averaging techniques. The literature (Basso 2008) also documents generic PWM model development using PSpice ABM constructs such as E and G Values and discrete components. These models are fast in simulation, but less sensitive/effective to transient analysis.



Top: PWM Controller UC3825 architecture

Bottom Left: Error amplifier compensation network

Bottom Right: Oscillator function

Figure 6-9 PWM controller UC3825/ SG1825 architecture and its main components

The main components of PWM controller which are relevant to this simulation work can be described with the aid of the UC3825 architecture shown in Figure 6-9. This architecture is applicable to SG1825 as well in terms of the main components and operation. The main components are the oscillator (OSC), error amplifier (E/A), PWM comparator and the output stage. Figure 6-9 also shows the hardware implementation of the fixing system the oscillator frequency which sets the inverter operating frequency.

In real hardware the oscillator frequency is set by the external components R_T and C_T. These components are respectively connected to R_T and C_T pins of the PWM controller as shown in Figure 6-9; oscillator function diagram. A nominal 3V appears at the R_T pin. The current flowing through component R_T is mirrored internally with a 1:1 ratio, which causes the identical current to flow out the C_T pin. This current flow charges the capacitor C_T and generates a linear ramp. Once the upper threshold is reached the discharge network reduces the capacitor potential to 1V, where a new ramp cycle begins. This process generates a triangular ramp signal V_M of peak-to-peak amplitude 2V (V_M swings between +3V (ramp peak) and +1V (ramp valley). This circuit is also responsible for generating a clock signal, CLK, which is used by the digital circuits.

The error amplifier is an operational amplifier, with a typical DC open-loop gain of 95 dB (60 dB for SG1825) and typical unity gain bandwidth of 5.5 MHz. This error amplifier can be gain-stabilised using conventional feedback techniques. The PWM comparator compares the error signal V_{ERROR}, produced by the error amplifier, with the ramp signal V_M generated by the R_T, C_T network. A 5V supply voltage for the error amplifier is internally generated. The PWM comparator output is clocked through a SR flip-flop and the associated digital electronics for glitch-free pulse operation. This prevents switching of the inverter by short pulses such as the

one encircled in Figure 6-8. Digital circuits also facilitate the generation of complementary signals that are required to produce a dual output. The output stage is a totem-pole output capable of delivering 2A peak current.

The chosen PWM macro model – SG1825 - is a mixed-signal PSpice model. In contrast to the real architecture of the SG1825, the model construct does not require external components R_T and C_T for generating V_M and CLK. An ideal voltage source - V_{RAMP} is employed for generating these signals. Model specific parameters "Period" and "Dead-time" are passed at the initialisation of simulation and using these parameters, V_{RAMP} produces V_M that poses the properties listed in Table 6-2. Parameter values 100 μs and 15μs have been used in most of the simulations for period and dead-time respectively. These values produce a 10 kHz switching signal with 85 % duty-cycle per arm, implying the inverter output would swing at 5 kHz, with the maximum dead-time of 15 μs between switching of each pair of IGBTs.

Description	Notation	value
V_M frequency	frequency	{1/period}
V_M ramp peak	V_{RAMP_PEAK}	3 V
V_M ramp valley	V_{RAMP_VALLEY}	1 V
V_M rise time	t_{rise}	{period - dead-time – 2*dead-time/100}
V_M pulse width	$t_{pulsewidth}$	{dead-time}
V_M fall time	t_{fall}	{dead-time/100}
V_M period	t_{period}	{period}

Table 6-2 PWM triangular ramp signal profile

The error amplifier open-loop gain and pole values are fixed in the model using E-Value gain and a discrete resistor, capacitor network. A reference on defining these values is documented in (Christophe 2008). The model produces an open-loop gain of 55 dB and unity-gain bandwidth of approximately 5 MHz (where gain crosses 0 dB gain). The low-frequency pole is placed around the 100 Hz region as the phase drops from 180°. Simulation results are

presented in Bode plot format, in Figure 6-10. This simulation result matches the manufacturer's data published in the SG1825 manufacturer's datasheet and this provides a validation of the model within the simulation. The output stage of the PWM is represented by two BJTs.

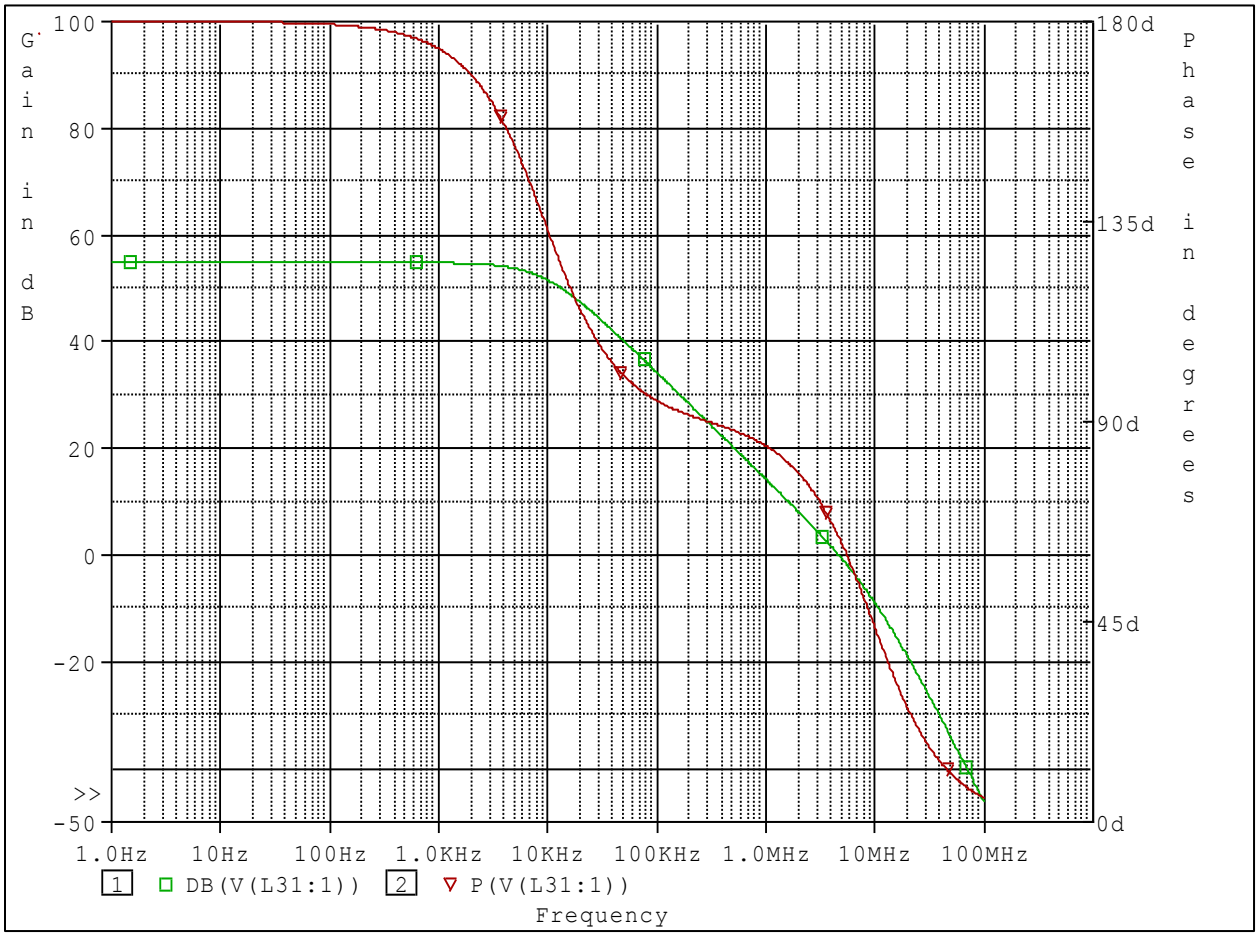


Figure 6-10 Open loop Bode plot of the SG1825 error amplifier PSpice model

Measurements on the PWM performance at component level were not available for evaluation. The stability of the system relies on the complete loop response. The loop includes the characteristics of compensation network, PWM gain and the power stage. Hence it is acceptable to study the stability scenario at a system level. In this Chapter validation of the component selection in the real hardware for achieving system stability is proved theoretically.

- Stability of the system

As there was no simulation of the system available to the HV power supply design and commissioning engineers, incremental adjustments were made by tuning the compensation circuit in the time domain until the stability of this system was achieved. This approach is justifiable as the characteristics of the power stage are unknown, and can vary from one module to other. Hence component selection by frequency response analysis of the complete system is not possible. For the same reason frequency response analysis was not used in this simulation work for the system model validity and system stability/ response analysis.

Transient/ time domain analysis begins by evaluating the transfer function of the compensation network built around the error amplifier; Figure 6-9: Error amplifier compensation circuit. The compensation network is served by the components R1, R2, R3, C1 and C2. The error amplifier, ERRAMP is internal to the PWM controller. The operation of the circuit can be expressed as follows: at DC (or in the low frequency of up to tens of Hz), C1 and C2 are open circuit, which makes the $VOUT_{ERRAMP}$ saturate to the ERRAMP supply voltage (~ +5V). At a higher frequency, both C1 and C2 are short circuited and the $VOUT_{ERRAMP}$ will operate as a simple inverter, with input resistance $(R1R2/R1 + R2)$ and feedback resistance R3, making the $VOUT_{ERRAMP}$ gain $(-R3(R1 + R2)) / (R1R2)$ At very high frequency, which is outside the bandwidth of the circuit, the $VOUT_{ERRAMP}$ distorts and become negligible.

With Figure 6-9; Error amplifier compensation network, as an aid, the transfer function – T(s) of the circuit can be driven as shown in Equation 6-3. Equation 6-4 was obtained by plugging the values R1 = 3.9 kΩ, R2 = 10 kΩ, R2 = 1.8 kΩ, C1 = 100 nF and C2 = 220 nF in Equation 6-3. These are the values of these components used in the real hardware. Equation 6-5

shows the transfer function in a simplified form. Solving the numerator N(S) and the denominator D(S) to 0, gives the system's zeros and poles. Figure 6-11, is an Argand diagram plotting the poles (p_1 and p_2) and the zeros (z_1 and z_2) for this particular system. Their values are as follows: z_1 = 114 Hz (719/ (2*π)), z_2 = 401 Hz (2.525 x 10^3/ (2*π)), p_1 = 0 Hz and p_2 = 407 Hz (2.56 x 10^3/ (2*π)).

$$T(s) = \frac{VOUT_{ERROUT}}{V_{IN}} = \frac{(SC_2R_3 + 1)(SC_1R_1 + SC_1R_2 + 1)}{SC_2(SC_1R_1R_2 + R_2)} \quad \textbf{Equation 6-3}$$

$$T(s) = \frac{VOUT_{ERROUT}}{V_{IN}} = \frac{5.5044 \times 10^{-7}S^2 + 1.786 \times 10^{-3}S + 1}{8.58 \times 10^{-7}S^2 + 2.2 \times 10^{-3}S + 0} \quad \textbf{Equation 6-4}$$

$$T(s) = \frac{VOUT_{ERROUT}}{V_{IN}} = \frac{(S + 719)(S + 2.525 \times 10^3)}{S(S + 2.56 \times 10^3)} = \frac{N(S)}{D(S)} \quad \textbf{Equation 6-5}$$

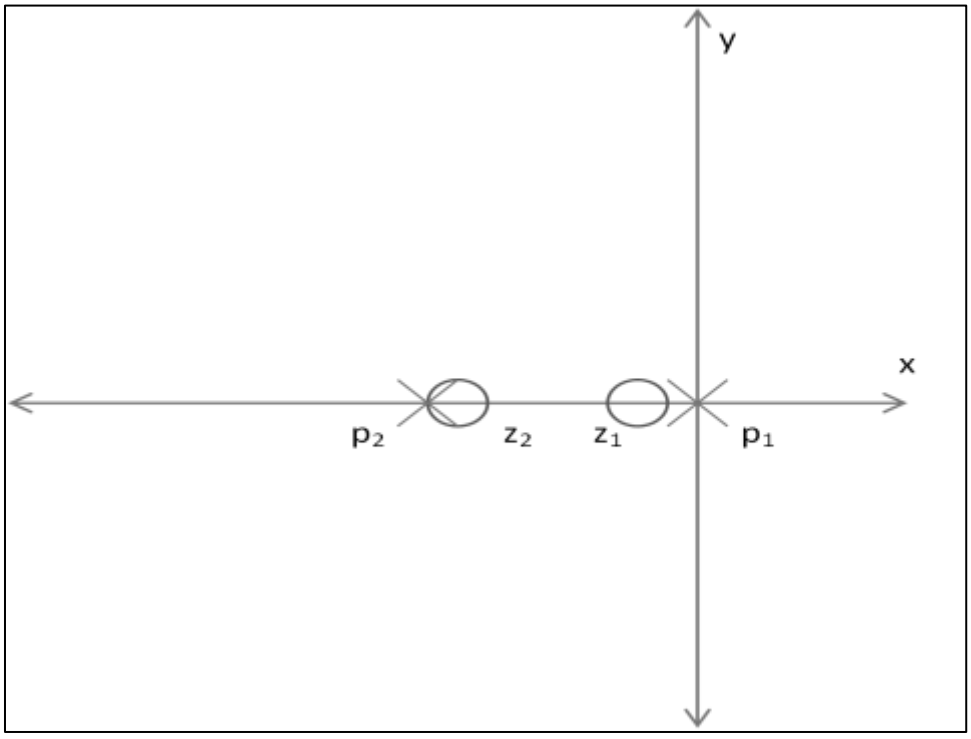


Figure 6-11 Pole-Zero diagram of the system

It is known that a system, whose transfer function has poles that can be plotted on the left hand side of an Argand diagram, is a stable system, which is bourne out by extensive operation of this system. However, what is not known is the transient response of the system nor the validity of it. The characteristic equation and its solution determine the form of the transient response for any input (Golten & Verwer 1991) and it is given by the denominator of the transfer function. In other words, the transient response can be predicted by the poles of the transfer function. This particular system has two poles p_1 and p_2 that are at the origin and at coordinates (-407, 0). The relationship between pole location and the transient term can be expressed as $T(t) = ke^{p_1 t}$ and $T(t) = ke^{p_2 t}$. T(t) yield a constant value of k for p_1 ($T(t) = ke^0 = k$) and a decaying response for pole p_2 as p_2 is a –Ve value. A pole at the origin does not have much influence on the transient response, but it sets the initial conditions of the system. A pole further away from the origin does significantly influence the transient response, however. Pole p_2 being a negative value means that the transient response is a decay response (e^{-407t}) and this exponential function decays down to approximately 0V, in duration 4 times its time constant given by $1/p_2$. This time is approximately 10ms.

Zeros in the system, affect the weighting of the transient behaviour collectively with the pole at the origin. However, they do not affect the response time. Further discussion on this aspect is covered in section 6.7.1, which includes further discussion and evaluation on the system stability and response time.

6.3.2 Fault detection and recovery

Faults that are detected by this circuit are caused by the load behaviour and its characteristics and these are known as flashovers. Flashovers present a short circuit to the high voltage that discharges the HV smoothing capacitor. The surge in output current from the

transformer to recharge the capacitor is apparent in a transformer drive current that exceeds a controlled threshold, and such events are termed ‘over current’ events. Over current events in the system are a potential threat to the semiconductor power switching devices, the high voltage transformer, and the EBW-Gun. Rapid recovery (recharging of the HV smoothing capacitor) is important otherwise the weld quality will also be jeopardised. Hence appropriate circuits are implemented in the system for the detection of the inverter load current and action upon it should it exceed the set overcurrent threshold.

A direct current transformer (DCT), with 5000:1 turns ratio is implemented in series with the high voltage transformer primary winding and one of the inverter’s output terminals, labelled as B in Figure 6-1. This DCT with an appropriate signal conditioning circuit generates a voltage signal that represents the inverter load current in a scale 500 A = 3.9V. This is the current feedback signal I_{FB}, and is compared with the overcurrent threshold setting by a voltage comparator. A fault signal – $V_{OVERCURRENT}$ is generated if I_{FB} exceeds the overcurrent threshold limit, which is processed by the second stage of the control circuit – the flashover detection circuit.

The flashover detection circuit, counts the number of times the fault signal $V_{OVERCURRENT}$ is asserted, and if it exceeds 10 times, then it flags an error signal $V_{FLASHOVER}$. A count of 10 overcurrent events is considered as the detection of a flashover having occurred. Digital circuitry, comprising a number of retriggerable monostable ICs with logic circuits provide this timing sequence. A count of 10 events also means 10 switching cycles of both pairs of switching devices (5 cycles each), lasting 1 ms. Note each half cycle is 100 μs duration for a 5 kHz inverter system.

The assertion of $V_{FLASHOVER}$ consequently makes the V_{KVOUT} demand value zero, which in essence dictates that inverter switching stops. A cloud of digital logic handles this by pulling the control signal V_{DEMAND} (Figure 6-7) to zero for 23 ms and ramping it back to its original settings over the next 7 ms, approximately.

6.4 Transformer modelling and inclusion of the saturation effect

The high voltage transformer is the final stage of the EBW power source's power stage. It steps up the inverter pulsed output V_{AB}, of voltage level 600 Vac@5 kHz to approximately 300 kVac@5 kHz. The construction specification of the high voltage transformer is as follows: The voltage ratio is 550/300kV. The turns ratio is 12/6660. Thus for $550V_{ac}$ input, the output is $305250V_{ac}$ (peak value). The 550V primary winding is 14 turns and it has a tap at turn 1 and turn 12. So, although we used 12 turns, tappings are provided for 11, 12, 13 & 14 turns if the output voltage needed some adjustment. The 300kV secondary is wound in 6 series connected sections. Each section is 1110 turns, wound at 160 turns per layer over 6 layers plus 1 layer of 150 turns. As calculated values, the leakage reactance is 26.4% (15.3uH referred to the primary) and the distributed capacitance of the secondary is about 3.37pF. The core is C core, 820 x 275mm window, 50 x 50mm section, CSA 22cm2 (using a 0.88 stacking factor), material Megaperm 40L (40%NiFe) from Vacuumschmelze in Germany. The core is assembled with a 0.13mm air gap in each leg. This manufacturer's data has been provided by the manufacturer – International Transformer (Johns 2012). The construct however handles 600 V_{ac}@5 kHz input and produce 333 kV_{ac}@5 kHz in reality.

The core saturation of the high voltage transformer contributes to the transient response of the system. Hence inclusion of an appropriate model of the transformer with a core saturation

effect is needed. A number of mathematical models that represents the nonlinear behaviour of the magnetic core have been published in (Committee, Working group C-5 of the systems protection subcommittee of the IEEE power system relaying 2000). One of the models discussed - Jiles-Atherton model, is incorporated in PSpice, for its accuracy in simulating the dynamic behaviour of magnetic devices. A detailed study of the model is outside the scope of this research work. In brief, this model is based on current physical theories of magnetic domains in ferromagnetic materials and is described in detail by (Jiles D, Thoelke J & Devine M 1992) and (Committee, Working group C-5 of the systems protection subcommittee of the IEEE power system relaying 2000). Inclusion of the saturation effects quickly hampers simulation time due to discontinuities and voltage glitches, which raises serious non-convergence issues (John et al. 1991). A more simplified model that uses passive components, together with E-Value elements in the ABM library was published by (Basso 2008) but it compromises accuracy for fast simulation, and a good tendency to converge. However, this simulation work uses the Jiles-Atherton model for better accuracy. The model requires six parameters to be inputted in the model template, which are listed below in Table 6-3.

The parameters were derived with the aid of data provided by the transformer manufacture (Johns 2012) and the magnetic core material datasheet (Vacuumschmelze 2012). The core dimension is pictorially presented in Figure 6-12. With reference to it, the path length (depicted with a broken line) can be calculated approximately (2 x (L+W) + 50 mm). With a stacking factor value of 0.88, the cross sectional area (CSA) can be calculated as 22 cm^2 (0.88 x 50 x 50 mm). The gap length is given by the transformer manufacturer. Values for the residual flux density (BR), saturated flux density (BM) and coercive magnetic force (HC) can be extracted from the core material datasheet. Simulation results presented in Figure 6-13 show the

B-H curve for the transformer core discussed. It reflects the aforementioned BM, BR, and HC values accurately. Note that the datasheet values are given in Tesla and A/cm have been converted to Gauss and Oersted respectively as per PSpice model requirements.

Parameter Name	Description	Value
LENGTH	Core path length	224 cm
AREA	Cross sectional area	22 cm^2
GAP	Core gap	0.26e-6
BR	Residual flux density	7500 Gauss
BM	Saturated flux density	14800 Gauss
HC	Coercive magnetic force	0.0753982 Oersted

Table 6-3 Transformer core parameters

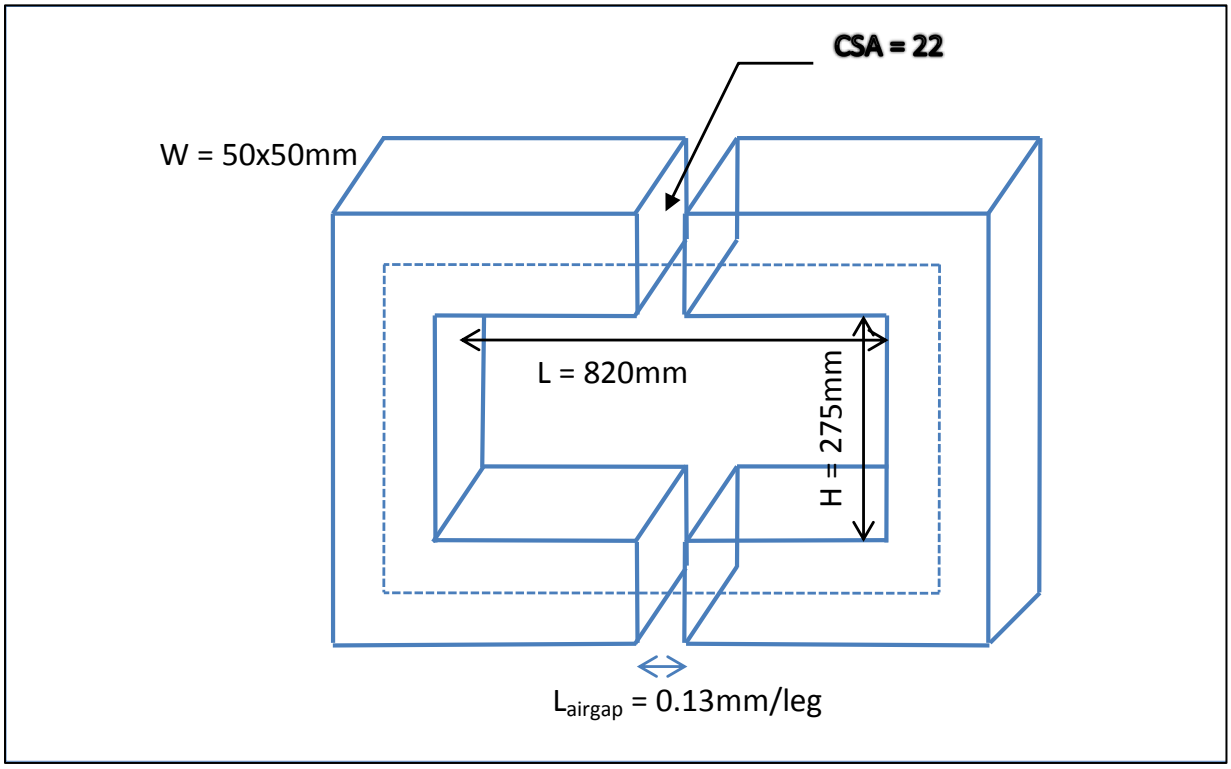


Figure 6-12 High voltage transformer core dimensions

The output stage of the inverter is connected to the HVT primary via an air cored choke - CHOKE1. The air core design avoids variation in choke inductance even at the overcurrent threshold. The choke has an inductance of 13 µH and acts to reduce *di/dt* during switching

current commutation into the HVT. It also protects the power stage during a short circuit. The saturation effect caused by the transformer core could not be verified at a component level. The preceding section 6.6.3, presents measurement results obtained at a system level highlighting the saturation effect on the inverter current.

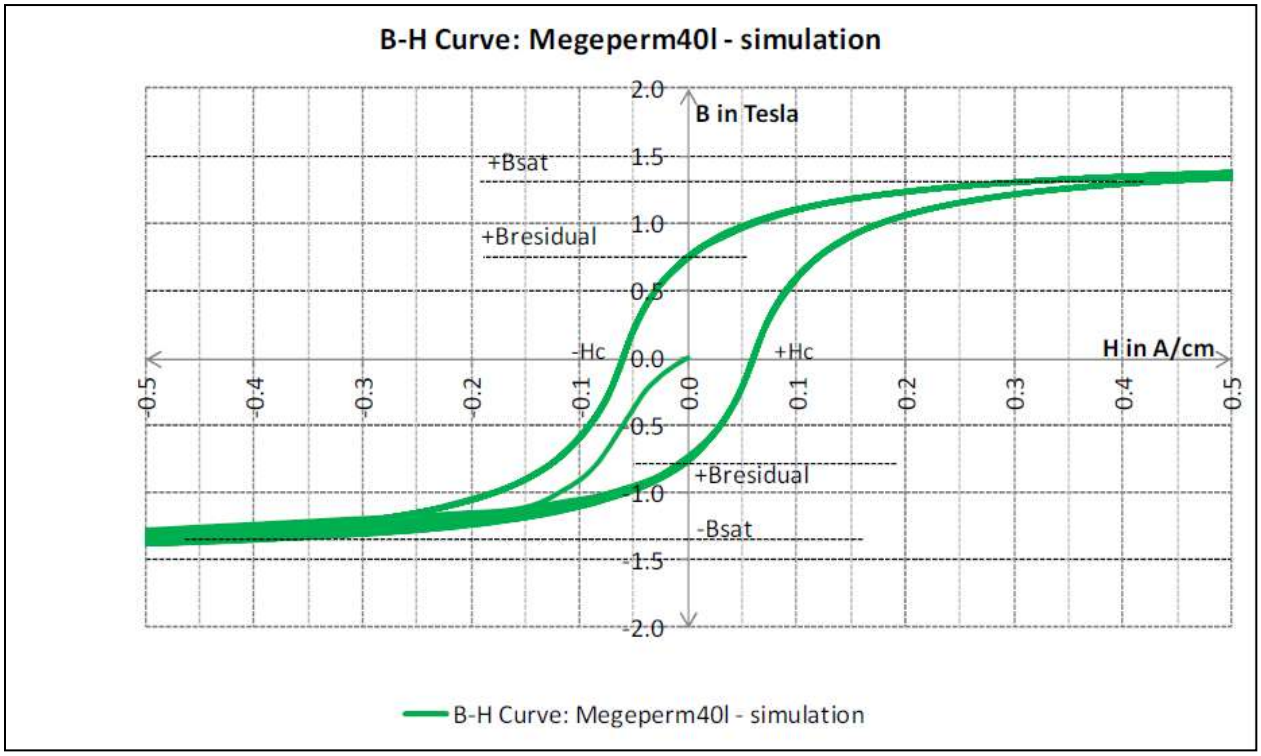


Figure 6-13 B-H curve of the High voltage transformer core

6.5 Load behaviour and modelling

The electron beam welding gun's behaviour is described as $I_{BEAM} = GV^{3/2}$, in the literature (O'neill 1959), where I_{BEAM} is the beam current generated when an accelerating voltage (V) is applied across the cathode and the anode of the EBW gun. The term G is a proportionality constant. A detailed study of the gun behaviour is outside the scope of this thesis. However, the inclusion of the gun's transient behaviour in the simulation is important as it is the load for the power source.

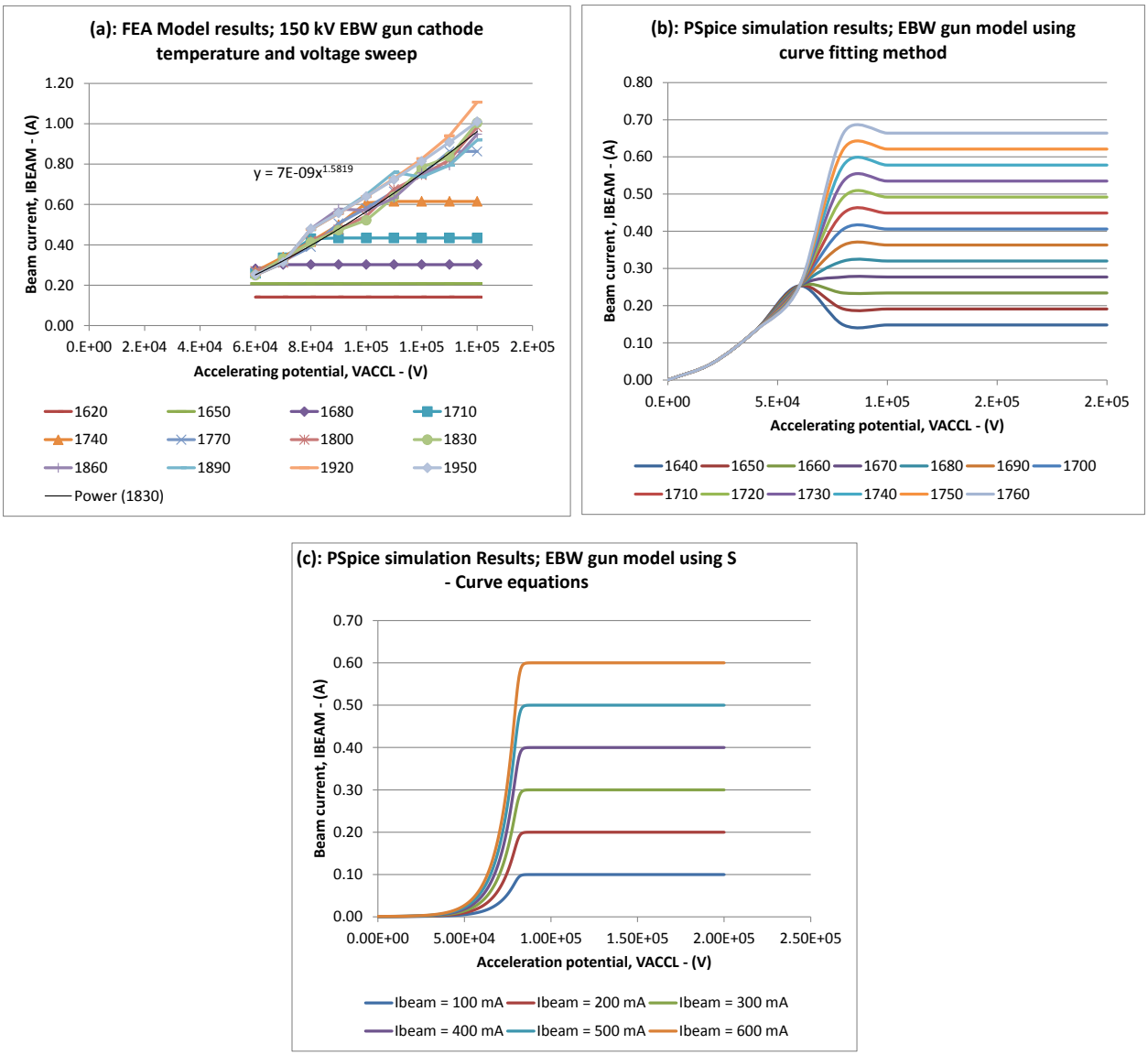


Figure 6-14 The I-V characteristics of an electron beam welding gun

The EBW gun is a thermionic diode and its behaviour can be compared to an electronic diode's forward biased characteristics. The gun is operated in the thermally limited state – that is to say the beam current is determined by the temperature of the cathode and is independent of the high voltage. The beam current is used to control the beam power (operation is normally at a fixed high acceleration potential). The temperature of the cathode is regulated by monitoring the beam current in a feedback circuit that adjusts the cathode heating power. The cathode is

indirectly heated, using RF coupled across the vacuum (Sanderson & Ribton 1998)The gun behaviour is shown in Figure 6-14; a, an FEA modelling results for a 150 kV, single stage EBW gun provided by the designer. The legends label the temperature of the cathode in Kelvin.

The modelling work firstly followed a curve fitting method applied to the FEA modelling results to find out equations for the trend of the curves. It then created a PSpice behavioural model that included nested mathematical formulae for reproducing the trend of the curves presented in Figure 6-14; a. The model simulation results are presented in Figure 6-14; b. Regardless of its accuracy, the simulation took a long time to converge due to the high power, and discontinuous mathematical equations. Hence, a different approach of using a simple formula for the S-Curve was followed. Simulation results for this equivalent EBW gun model are presented in Figure 6-14; c. The simulation results obtained for this model follow a similar trend to that described in the literature and the FEA model results. In addition, the simulation time was greatly reduced.

6.6 Top level simulation and analysis

Having produced simulation models for relevant subsystems and their critical components and/ or functionalities, the developed models were stitched together, to form a system level model that allows transient simulations to be carried out in a PSpice environment. The architecture of the developed top level system model is shown in Figure 6-15. All models created are included in Appendix A for reference.

Transient analysis in PSpice computes the time response of the circuit, taking into account all the nonlinearities of the circuits. The integration of all the developed components and subsystems to form a top level system model makes the circuit large, and makes the simulation

computation time longer. Besides this, the inclusion of charge components such as capacitors and inductors, make the simulation struggle to compute the DC solution, which is the preceding step to transient analysis. Moreover the integration of subsystems whose functionality differs from the other subsystems makes the transistors in the circuits have different operating points, make it challenging for the simulation tool to find solutions.

To overcome these issues, to make the large circuit simulate, in addition to making the models simple, and relaxing the simulation option settings, discrete components such as resistors and inductors were added external to the developed models. These external components make simulation converge by limiting *di/dt*, *dv/dt* and prevent short circuit conditions (in capacitors) and open circuits conditions (inductors) at initialisation. This remedy does not compromise the simulation results as even in real systems these components exist in the form of parasitic values. These add on components are highlighted in Figure 6-15 as "auxiliaries". Other instants are discussed previously in relevant sections.

The functionality of the system model is explained in flow chart format in Figure 6-16 and the step numbers correlate with the step numbers given to the subsystems in Figure 6-15. In essence, when a certain acceleration voltage is demanded the PWM starts to output switching pulses with a duty-cycle dictated by the error signal level. The IGBTs are turned on via the IGBT drive circuits appropriately, to provide a $600V_{ac}$ at 5 kHz at the input of the step-up transformer input. The step-up transformer together with the rectification circuit provides the required acceleration voltage. The feedback circuits regulate the voltage output. Fault recovery circuits act upon overcurrent events.

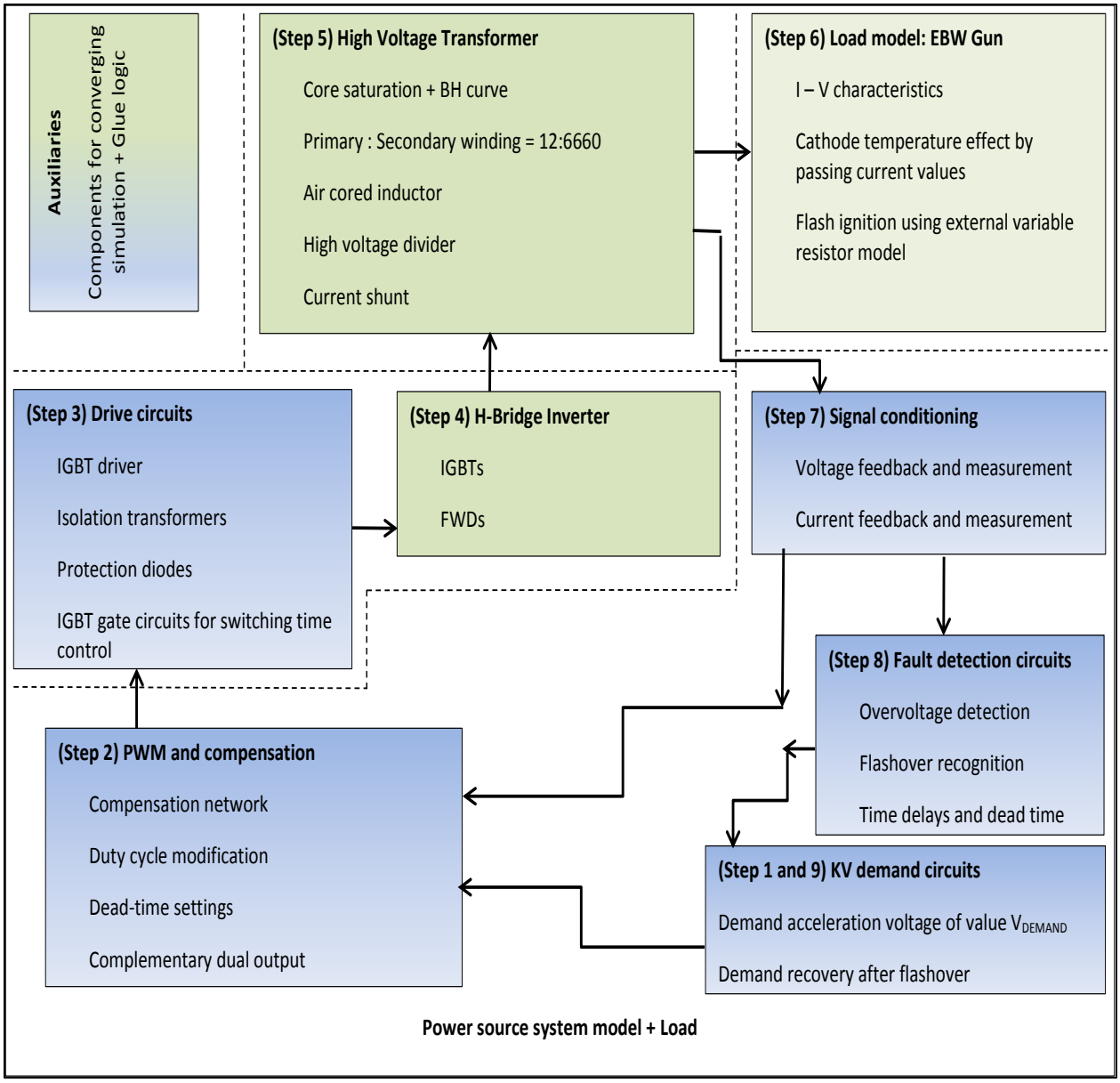


Figure 6-15 Top level stitching of subsystems

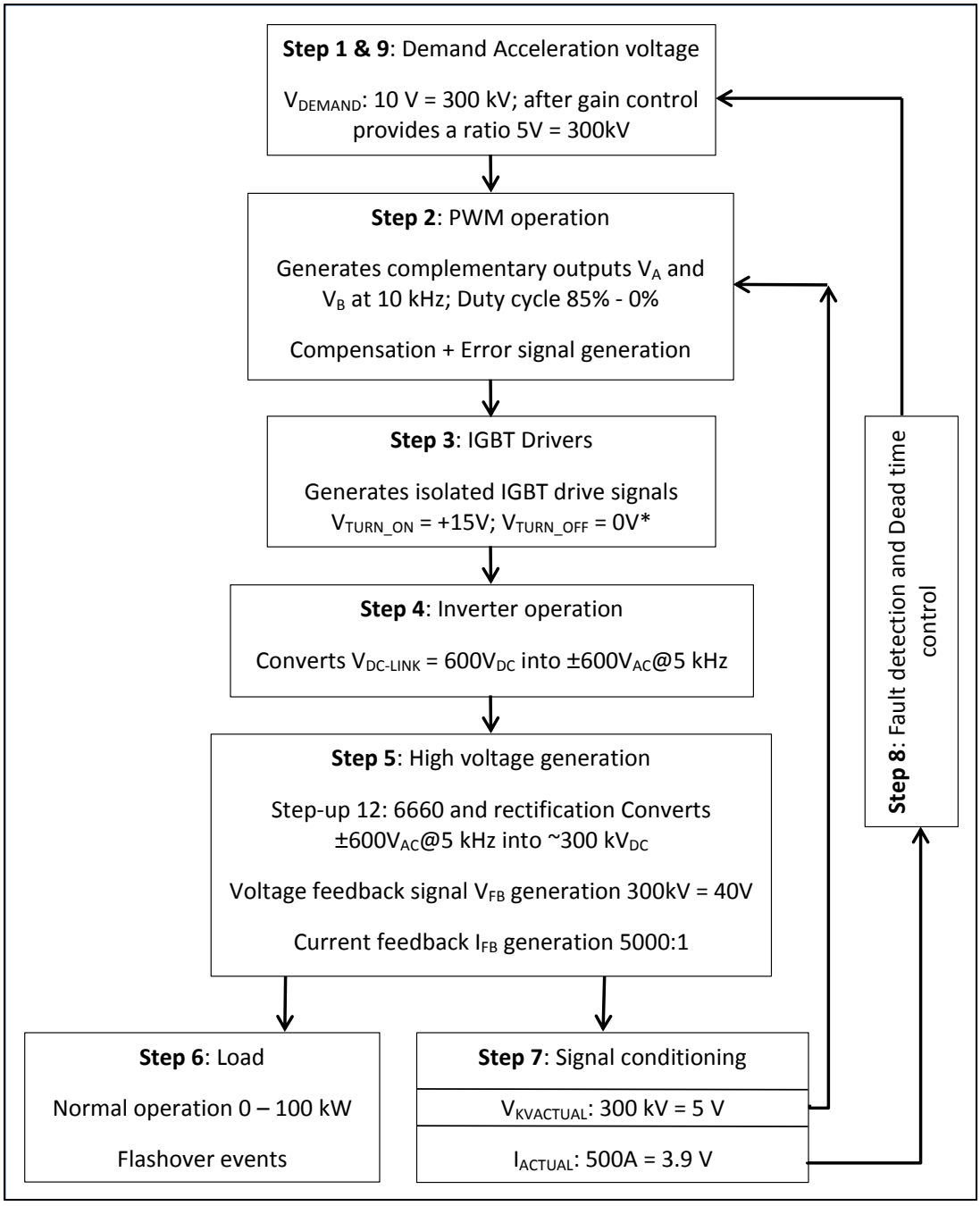


Figure 6-16 EBW Power source top level simulation model functionality

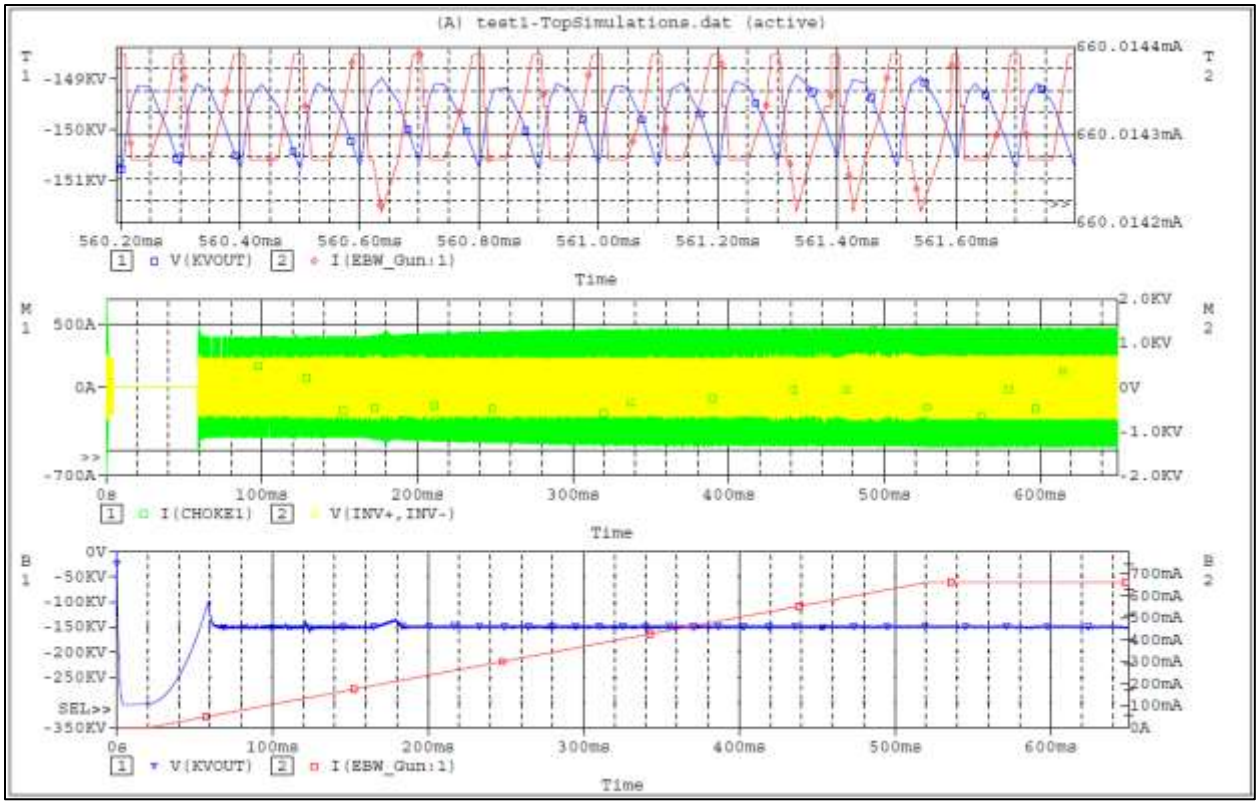


Figure 6-17 Simulation showing a typical flashover free 150 kV; 600 mA power delivery

Transient simulation of the completed system model for the power source, showing a typical flashover free operation, is presented in Figure 6-17. The system model was simulated for the simulation conditions; V_{KVOUT} = -150 kV; I_{EBW_Gun} = -660 mA with a ramp up time 500 ms, starting from 10 ms. The power source voltage output V_{KVOUT}, ramps up initially, and charges beyond the demanded 150 kV. This is shown in Figure 6-17; B1. This is because of the system's settling time. In a previous section 6.3.1, the theoretical settling time was discussed as approximately 10 ms. In this particular simulation, the settling time is much longer at 60 ms for the following reasons; as the power source is not loaded, i.e., no load current (I_{EBW_Gun}) is demanded until t = 10 ms as can be seen in Figure 6-17; B2, the output stage is charged to its maximum (~300 kV) in the short duration of less than 2 ms. Note that the inverter stopped switching at this stage, stopped delivering current (Figure 6-17; M1 - I_{CHOKE1}) and voltage

(Figure 6-17; M2 – $V_{IN+,IN-}$). Inverter switching resumed as significant power from the power source is demanded when delivering 660 mA at a regulated 150 kV.

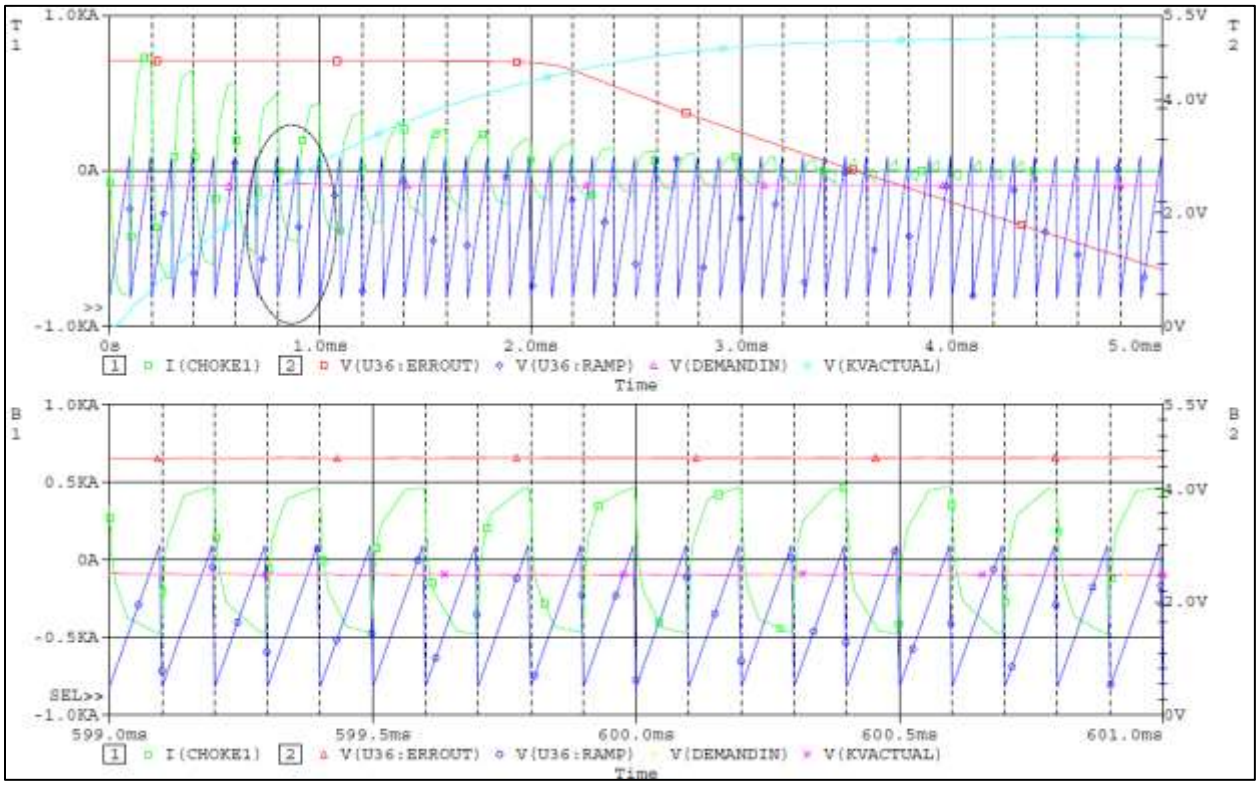


Figure 6-18 Inverter current for light and heavy load

Observation and lessons that can be learnt from this set of simulation results are significant as listed below:

- The primary current to the high voltage transformer is the inverter output current, I_{CHOKE1}. For delivering 660 mA load current- I_{EBW_Gun}, I_{CHOKE1} is as high as 500A implying the primary resistance at transformer saturation can be as low as 1.2 Ω (600V / 500 A). This is collectively the parasitic primary resistance and the reflected impedance from the secondary side of the transformer. In this particular simulation the primary parasitic impedance was set

to 0.7 Ω using an ideal resistor model. At light load, I_{CHOKE1} can be as low as 200 A. This is highlighted in Figure 6-18; B1 and T1, respectively.

- The figure also shows the slow response of the compensation circuit, revealing even that 150 kV is reached at 0.8 ms (encircled where $V_{KVACTUAL}$ crosses $V_{DEMANDIN}$) the error signal does not start to respond until 2.2 ms when it starts ramping down slowly.
- It also can be noticed that the shark-fin shape of I(CHOKE1), exposes two different time constants, highlighting the saturation effect.

6.6.1 Output ripple analysis

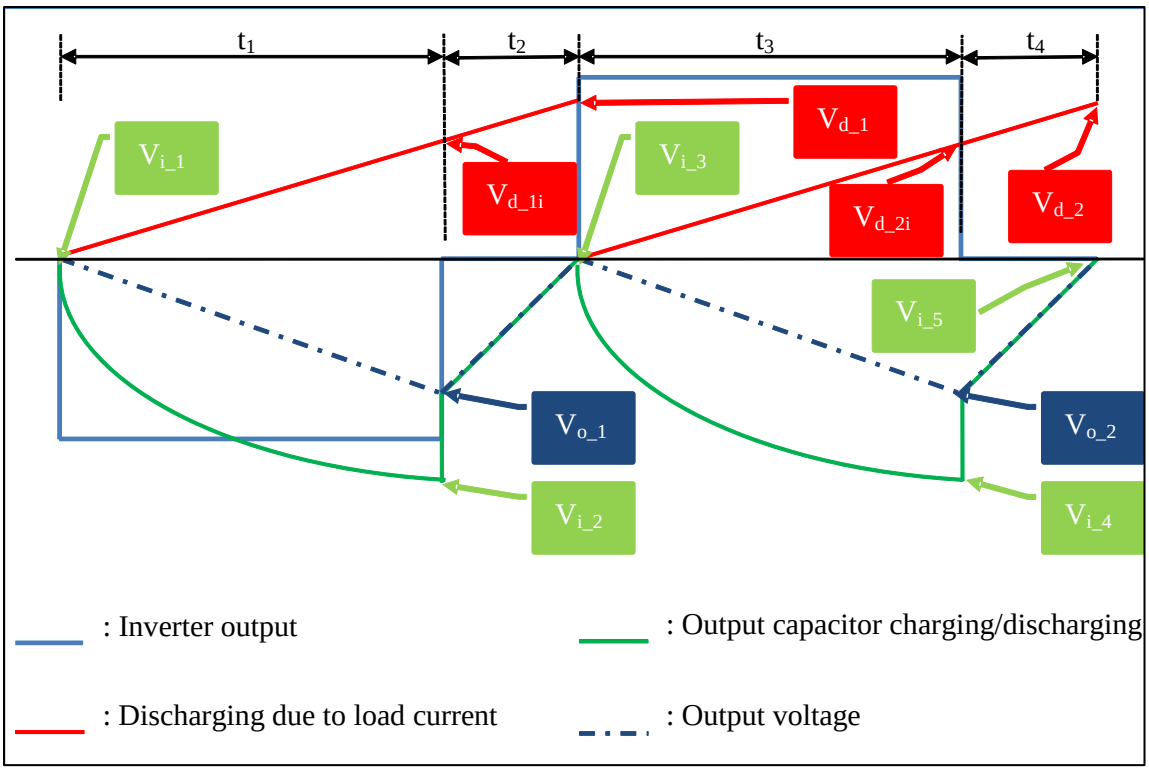


Figure 6-19 Acceleration voltage V_{KVOUT} ripple explained (Note: Not drawn to scale)

The acceptable ripple on output voltage V_{KVOUT} and current delivered to the load I_{EBW_GUN} are 2% (peak-to-peak) and 5% (peak-to-peak) respectively according to the British standards for EBW machines (BS EN ISO 14744-1 2008). However precision welding

applications prefer a much lower ripple. The ripple seen on a typical 150 kV; 660 mA simulations are included in Figure 6-17; T1 and T2, for V_{KVOUT} and I_{EBW_Gun} respectively. A voltage ripple as high as 1.3 % ($100 \times (151\,kV - 149\,kV)/150\,kV$) has been observed. The current ripple noticed was insignificant, as the beam current is independent of the high voltage level in the thermally limited case. Justification for the voltage ripple can be addressed by a theoretical method, as discussed below with the aid of Figure 6-19.

The EBW power source, has a 4 nF capacitor at its output as a smoothing/ reservoir capacitor. The 10 metre high voltage cable that connects the anode of the EBW-Gun to the power source output has a cable capacitance of value 100 pF/m, making the total effective capacitance at the output of the power source 5 nF. Based on providing 150 kV as V_{KVOUT} and drawing 660 mA as load current I_{EBW_Gun}, the expected ripple on V_{KVOUT} can be calculated as follows:

The charging characteristics of the output capacitor are given by Equation 6-6. Parameters V_C, V_F, V_i, and T are the capacitor voltage after charge time t, the final value of voltage, the instantaneous initial value of voltage of the capacitor, and the time constant respectively. The time constant T is the product of the equivalent output capacitance (C_{OUT}) and the load resistance (R_{OUT}). Measuring the value of R_{OUT} is complex and for this calculation R_{OUT} is assumed to be 227 kΩ (V_{KVOUT}/I_{EBW_Gun}). Parameter t is the charging time. Assuming the switching duty cycle is at its optimum 85%, the charging time of the capacitor is approximately 85 μs. This is indicated as t_1 and t_3 in Figure 6-19. Labels t_2 and t_4 are the dead time during which the capacitors do not get further charged.

$$\mathbf{V_c = V_F + (V_i - V_F)e^{\frac{-t}{T}}} \qquad \textbf{Equation 6-6}$$

$$\mathbf{V_{discharge} = I_{EBW\_Gun} \times t/C} \qquad \textbf{Equation 6-7}$$

Voltage	Process	Formula	Calculated Value
V_{i_1}			-150 kV
V_{i_2}	Charging	$= -333\ kV + (-150\ kV + 333\ kV) \times e^{-85\mu s/1.135ms}$	-163.204 kV
V_{d_1i}	Discharging	$= 660\ mA \times 85\ \mu s/5\ nF$	-11.220 kV
V_{o_1}		$= V_{i\_2} - V_{d\_1i}$	-151.984 kV
V_{d_1}	Discharging	$= 660\ mA \times 15\ \mu s/5\ nF$	-1.980 kV
V_{i_3}		$= V_{o\_1} - V_{d\_1}$	150.004 kV
V_{i_4}	Charging	$= -333\ kV + (-150\ kV + 333\ kV) \times e^{-85\mu s/1.135ms}$	-163.204 kV
V_{d_2i}	Discharging	$= 660\ mA \times 85\ \mu s/5\ nF$	-11.220 kV
V_{o_2}		$= V_{i\_2} - V_{d\_1i}$	-151.984 kV
V_{d_2}	Discharging	$= 660\ mA \times 15\ \mu s/5\ nF$	-1.980 kV
V_{i_5}		$= V_{o\_1} - V_{d\_1}$	150.004 kV

Table 6-4 Output voltage ripple calculation

The lable V_{i_1} is the value of V_i at the start of time t_1 and it is assumed to be 150 kV indicating the demanded V_{KVOUT} is reached. V_F is the final value to be reached and it is calculated as 333 kV. This calculation is based on the high voltage transformer ratio of 12:6660, stepping up the inverter output V_{AB} of value ± 600V. Voltage drops across the rectifiers are ignored for simplicity. Using Equation 6-6, the charging values of the output capacitor can be approximately calculated. The discharging is assumed, due to the load current $I_{EBW\_Gun,}$ to occur for the time period 100 µs (switching time 85 µs and dead-time 15 µs) only for simplicity. The voltage drop due to discharging is calculated using Equation 6-7. Value C is the output capacitance 5 nF. The calculated values for the

labelled points are presented in Table 6-4. The rows highlighted form the trend of the output voltage, V_{KVOUT}, which has a ripple of approximately 2 kV.

6.6.2 Fault detection and recovery

The system model of the power source includes the fault detection and fault recovery control circuits. Their operation is discussed in section 6.3.2. In essence the monitored AC inverter current I_{CHOKE1} is precision rectified and compared with the overcurrent threshold level, typically 600A. The resultant signal, referred to as $V_{HTDEADSET}$, limits the output voltage V_{KVOUT} to 0V if the protection limit is reached for more than 10 cycles (typically, over 1 ms duration). Simulation results obtained showed a rectified value of 3.89V for I_{CHOKE1} = 498 A (almost equal to 3.9 V = 500 A) in Figure 6-20; ($V_{C5:2}$) as expected. The precision rectifier circuit implemented in the design allowed this accurate measurement. In this simulation the protection level is set to ~ 650A by setting the threshold to ~ 5.1 V as shown in Figure 6-20; $V_{R3:2}$ as set in the real hardware. This simulation results closely matched the design calculation, proving the fitness of the equivalent circuit model developed for the current transformer (current shunt) and the accurate performance of the topology specific models used to represent the overcurrent detection circuits.

The complete operation of fault detection and recovery was simulated with the default 30 ms Weld Dead Time, after flashover detection. The simulation results are presented in Figure 6-21 and are used as an aid to discuss circuit operation. The power stage output was short circuited at simulation time 2 ms to emulate the flashover fault condition. A simple variable resistor model, modelled using a control source G in the PSpice ABM library was used to simulate this condition. The power stage output was connected to ground via this model resistor and its value was controlled using a pulse voltage source with pulse profile: Initial value = 75

MΩ; time delay = 2 ms; final value = 20 kΩ; fall time = rise time = 1 ms; and pulse width = 2 ms. This allowed simulating a leakage current of 2 mA @ 150 kV which is experienced in real hardware and a flashover current of ~13 A for 2 ms with a rise and fall time of 1 ms at a simulation time of 2 ms. A flashover current of 13 A is an arbitrary high current value that demands very high current from the inverter, making the transformer saturate and causing an overcurrent situation.

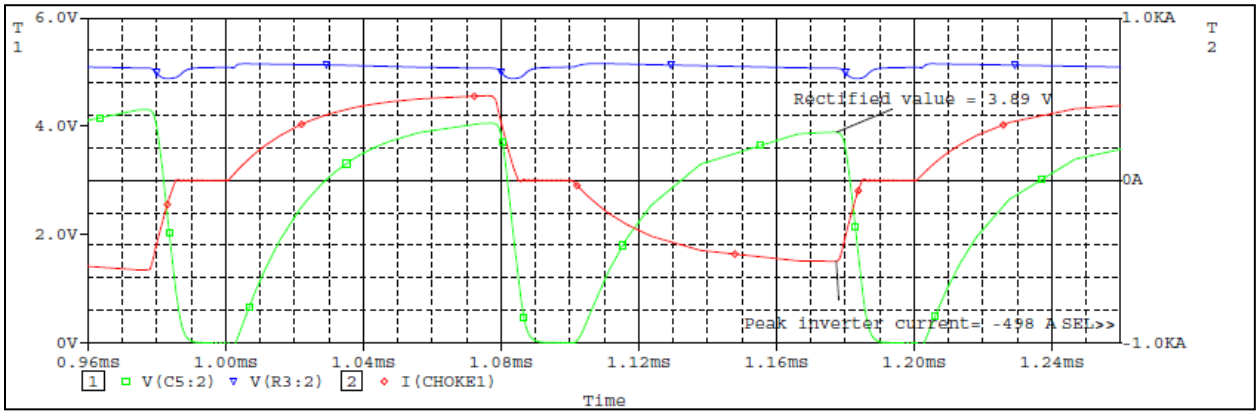


Figure 6-20 Conversion of inverter output current into DC for overcurrent detection (simulation)

On detection of flashover, an error flag $V_{HTDEADSET}$ is generated, which consequently forces the $V_{DEMANDIN}$ to 0V, which in turn makes the acceleration voltage V_{KVOUT} 0 V. This operation maintains the V_{DEMAND} at 0 volts for 23 ms. This time is set in the hardware using a fixed RC network. At the end of 23 ms, the V_{DEMAND} is resumed back to its original setting, 2.5 V in this particular simulation, demanding 150 kV output. The ramp up time is ~ 7 ms set by the hardware at present. As can be seen the V_{KVOUT} resumed back to 150 kV with minimal overshoot (compared to the initial stage) just after the V_{DEMAND} recovery.

Traces included in Figure 6-21; M1, highlight the V_{CE} voltage across the IGBTs. This is of interest to see if the V_{CE} overshoots above the IGBT breakdown voltage of 1200 V during

recovery. At recovery, the IGBTs go under a hard-switching operation switching approximately 600 A at 600 V and maximum duty cycle of 85%. Simulation reveals safe operation is maintained during the recovery operation, as the V_{CE} is kept below 800 V.

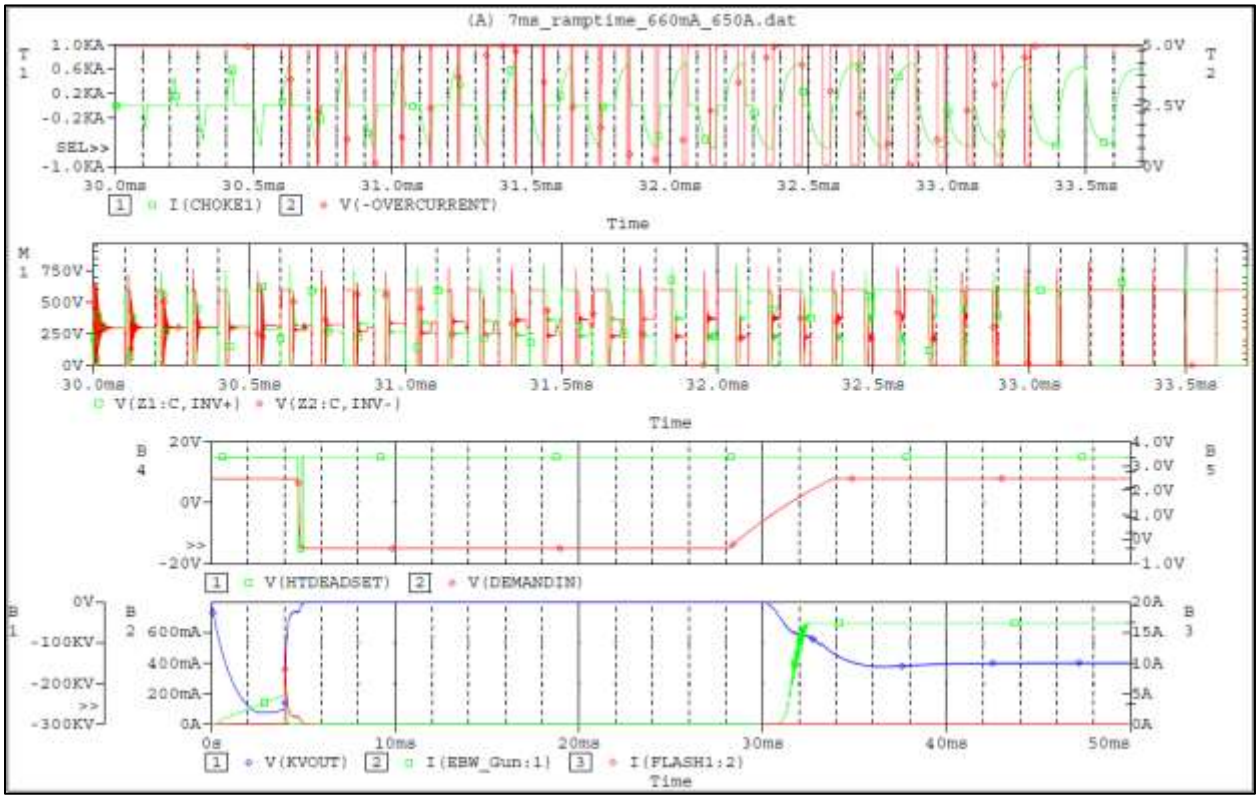


Figure 6-21 Simulation showing flashover detection and V_{KVOUT} recovery in 30 ms

6.6.3 Evaluation of system level operation

Real hardware measurements at a component level suffered because of the system complexity and limited resources. So the evaluation of all the developed independent component models was impossible. Efforts had been made to obtain inverter related measurements at a system level. This is because as aforementioned the EBW systems functionality heavily depends on the inverter operation and the control circuits implemented in effect controls the switching of the inverter. Hence a compromise had been made, to agree the simulation model is acceptable, if the inverter simulation results closely map the real hardware measurements. As an additional

measure to maintain the validity of the model, all important control electronics subsystems were modelled using topology specific models. Topology specific models are considered to be accurate and they closely map real hardware.

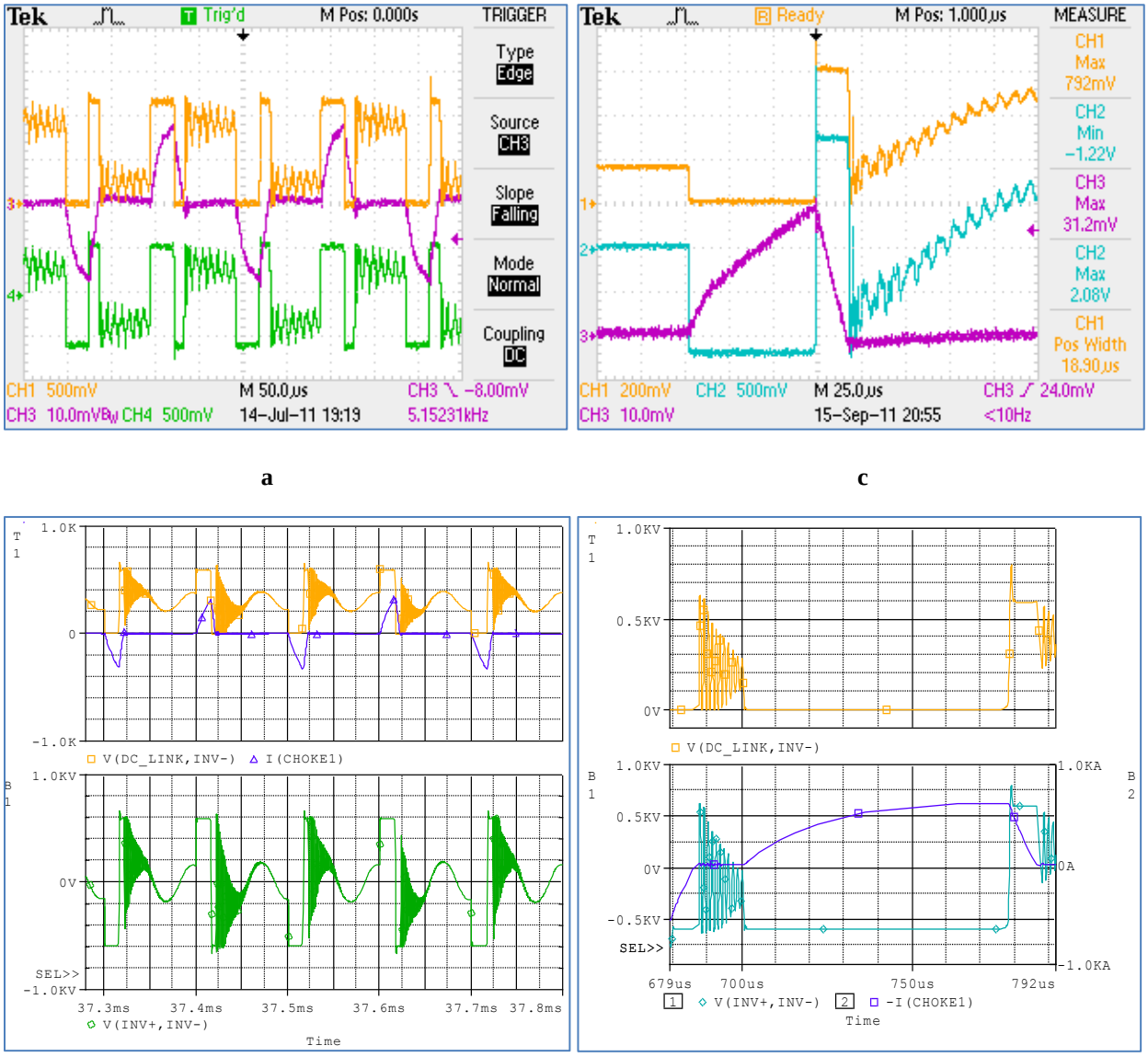


Figure 6-22 Inverter performance: figures a and c show the measurement on real inverter hardware depicting inverter output +/-610V and one of the IGBTs switching 300A and 600 A respectively. Figures b and d shows the corresponding waveforms in simulation for 300 A an 600 A respectively.

Modelling and evaluation of the power switching device IGBT - DIM800DDM12-A000 utilised in the inverter design is presented in section 6.2.2. These devices experience high di/dt when used in inverter applications. As a consequence, the IGBTs experience a high voltage stress as discussed previously due to the presence of parasitic inductance. Keeping the V_{CE} of IGBTs below their breakdown voltage (V_{BR}) is paramount and the FWDs integrated in the CO-PACK protects against V_{BR} violation when the devices switch off. The operation of the inverter was simulated, and compared with the measurements for validation.

Figure 6-22 presents the behaviour of the inverter at two different duty-cycles. Figure 6-22; a, shows a 25µs turn on of IGBTs. Commutation of the load current (CH3) of value 300A and turning it off at high di/dt is highlighted. The V_{CE} across one of the IGBT approaching ~ 800 V was noticed (CH1) during turnoff. Note that $V_{CE} = V_{DC\_LINK} + V_{SPIKE}$. The DC_Link voltage V_{DC_LINK} was measured as 610 VDC, and varies depending upon the supplied 3 phase voltage by up to 16%. The inverter output swinging between +600VDC and -600VDC is also shown (CH4). This is a typical operation of the inverter delivering ~ 20 kW power to load (150 kV@150 mA). The same scenario was simulated and the corresponding results are presented in Figure 6-22; b. The load current, V_{CE} and the inverter output are named as I_{CHOKE1}, $V_{DC\_LINK,INV-}$ and $V_{INV+,INV-}$ respectively. Close mapping of the simulation results to the real hardware was observed. Similarly measurements obtained for 600A load current commutation, and the simulation results obtained for 600 A are presented in Figure 6-22; c and d, respectively. In both cases the turn-on time was 75 µs.

6.7 Applications

The close mapping of the simulation results to the real hardware measurements and theoretical calculations validated the use of the system model, which could then be used for finding optimised parameters, to improve system functionality. Two such applications that require attention are discussed.

6.7.1 System stability

An exercise has been conducted to see the overshoot behaviour of the output voltage V_{KVOUT}, for a different configuration of the compensation circuit discussed in section 6.3.1. As stated, the overshoot behaviour (which is the weighting of the transient response) can be tuned by adjusting the zeros of the transfer function.

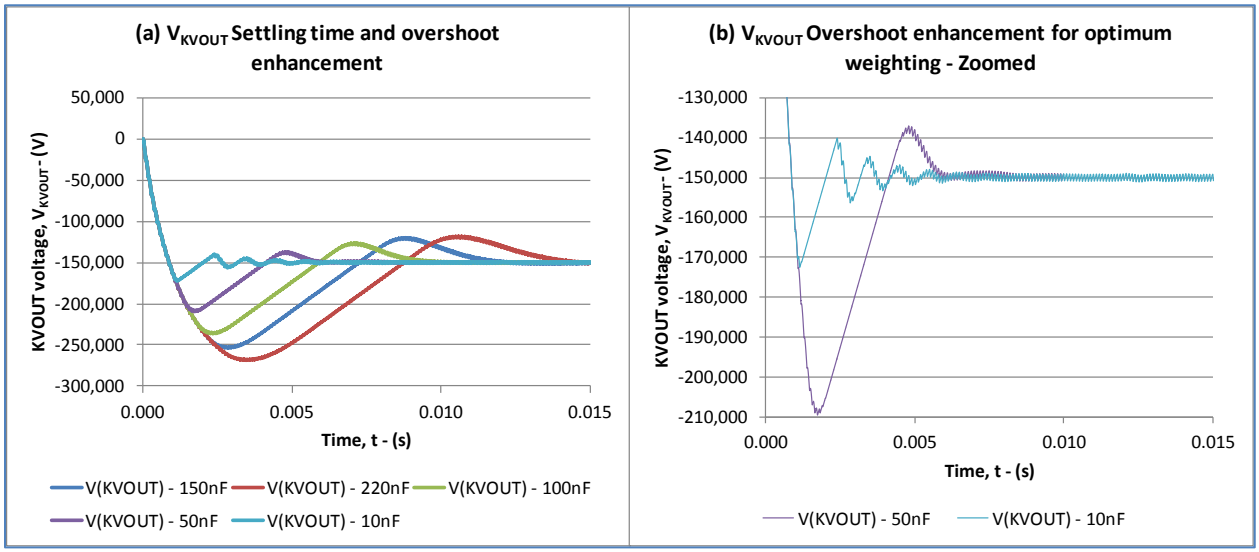


Figure 6-23 Stability of the system

The zeros of the transfer function T(S) of this system can be adjusted by changing the value of C2 in the compensation network, without compromising the settling time. Simulation was carried for analysing the effect of tuning the weighting of the transfer function by

parameterising the capacitor value C2, ranging from 10 nF to 220 nF. These transient simulation results are presented in Figure 6-23; a, for varying C2 values. It shows the transient behaviour of V_{KVOUT} during the initial start-up at which a 150 kV is demanded as a step. V_{KVOUT} stabilised to its final value of 150 kV for all the simulations, with the smallest overshoot for a C2 value of 10 nF. The maximum overshoot was obtained for a C2 value of 220 nF. Note that the real hardware is fitted with 220 nF for C2. The transient response time did not change due to this change in weighting. Figure 6-23; b, is a zoomed version of Figure 6-23; a, highlighting the transient behaviour of the system for C2 values 50 nF and 10 nF, which also require about 10 ms to settle down.

The advantage of using modelling to simulate the response of the power supply to changes in the feedback system operation is that the risk of damage – caused by overvoltage and high voltage breakdown in the transformer or other HV components is avoided. Instability in HV feedback loops poses a high risk to expensive, bespoke made HV components. Where experience exists (from the commissioning stage of power supply development) there is a good correlation between practical results and the model.

The theoretical analysis, in section 6.3.1 on system stability, concluded the response time, can be as high as 10 ms which is reflected in the system level simulation.

6.7.2 System modelling fault recovery control circuit enhancement

Enhancement of the control circuit that protects the IGBTs in the inverter against overcurrent, when large current transients or excursions occur as happens during micro-discharges and flashover, is crucial for long lasting operation of the system and defect free welding. Reduction in the duration of the weld dead time after flashover has been identified as a

parameter that requires optimisation. This is because it was revealed in the literature study (section 5.3.5) that the weld dead time period of 30 ms, set in the hardware is well above the operational constraints. The weld dead time includes 23 ms of 0V V_{DEMAND} time ($T_{DEADTIME}$) and 7 ms of V_{DEMAND} recovery time (T_{RESUME}). The time $T_{DEADTIME}$ as discussed in section 5.3.5 relies on the physical properties of vacuum integrity recovery. Shortening of this time relies on a knowledge of arc quenching and vacuum recovery, which is unknown at present. However, the time T_{RESUME} can be shortened if the power system and its control circuits can handle the rapid recovery.

Exercising this in the real hardware would carry a high risk for the inverter IGBT components and associated electronics, but with the aid of the simulation model this exercise was carried out for various settings and scenarios. One such exercise is included here, showing that the T_{RESUME} time is reduced from the default 7 ms to values 5ms, 3 ms, 1 ms and 500 μs. The response is depicted in Figure 6-24. As can be seen the system resumed to its original weld setting for all values, but collapses for a T_{RESUME} value of 500 μs. Stress analyses on the components, especially on the IGBTs, were also carried out. Appendix B includes the results. Justification for the collapse of the system transient can be analysed with approximations as follows:

Considering 150 kV; 660 mA power delivery, the inductor current I_{CHOKE1} flowing through the high voltage transformer's primary circuit is approximately 500 A. This is shown in previous sections. With the turns ratio of 12: 6660, the secondary current flowing through the smoothing and cable capacitor (4 nF and 1 nF respectively) can be calculated as approximately 0.9A ($500\,A \times 12/6660$). Plugging values for a capacitor value C = 5 nF, a charging current I = 0.9A and a charging time t = (n x 85 μs), where n represents number of cycles in the capacitor

charge Equation 6-8, reveals a value for n ≅ 10. Each cycle in this operation is 100 µs duration, totalling 1 ms for 10 cycles. This implies, the charging of the output capacitors, to 150 kV requires a minimum of 1 ms.

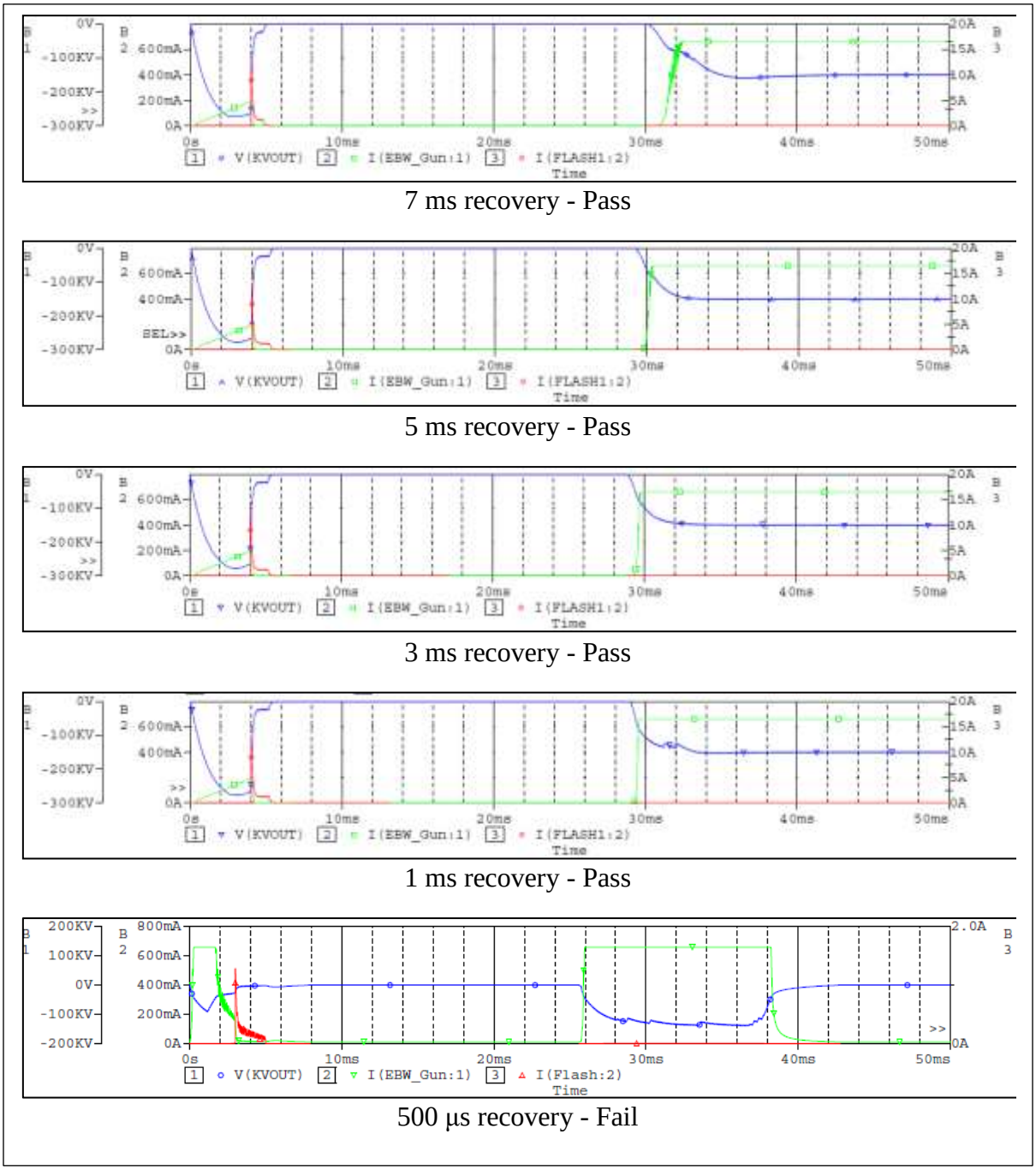


Figure 6-24 Fault recovery time optimisation

$$Q = CV = It$$

Equation 6-8

6.8 Summary

An Electron Beam Welding power source was analysed for implementing hardware changes for performance enhancement. The system modelling approach allowed the system to be investigated at many levels which was impossible with the real hardware due to its complexity. The model was constructed from a number of accurate subsystems models designed to derives theoretical values and manufacturers' supplied data. There was found to be close mapping of the simulation and real hardware output results. The simulation results obtained revealed the possibility of enhancing the HV power supply fault recovery control circuit to provide better performance during gun flashovers for defect-free welding. It also revealed the possibility of enhancing the transient behaviour of the system by modifying the system stability and characterisation circuit. In both cases the status of components or circuits implemented in the design were not compromised.

7 CONCLUSIONS AND RECOMMENDATIONS FOR FURTHER WORK

Design verification of mixed-technology and mixed-signal systems can be difficult and complex. This research helped advance the assessment procedures to reliably and systematically assess hardware design changes for two such systems – a LRUT pulser-receiver and a EBW power source, by means of integrated system modelling. The process addressed the utilisation of component models with different abstraction levels and simulates them in a computing environment to answer system and component level questions, verify system performance and reveal problematic interactions. SPICE based simulation tools were used for modelling and simulating the model constructs for its inherent ability to support modelling and simulating electronic circuits.

The research studied the feasibility of enhancing the portability of the pulser receiver in terms of its size and weight. Portability enhancement by 33% was achieved in terms of size and weight by downgrading the specification of its primary power source – a battery. The extension of this work facilitated enhancing the existing pulser receiver hardware to an experiment enhanced LRUT application of inspecting complex components. The research also critically analysed EBW power source hardware for obtaining optimum operating parameters that can be implemented in the hardware for enhancing a weld recovery sequence after major flashover events. Operating parameters, which shortens the weld recovery sequence by 6 ms had been identified which is anticipated to improve weld quality. Component values which help improve the transient overshoot of the acceleration voltage were also derived for enhancing the system performance.

The motivations for the formalisation of the LRUT subsystem models were manifold, ranging from proof of concept, validation of hardware specification and design modifications, verification of the correctness of the design, understanding and communicating the functionality of the design at a component level as well as at a system level, synthesis of circuits and as a test platform for maintenance tests. The models facilitated: a feasibility study on the system's portability enhancement; verifying new and existing designs, schematic generation and rapid prototyping of the system's power supply circuits; and enhancing the hardware to experiment an enhanced LRUT application. This system is a multi-technology system with integrated mixed-signal circuits. Combination of top-down and bottom-up methodologies were followed in the development of the system model for including relevant information and to achieve complete simulation. The simulation tool LTSpice was utilised for modelling and simulating the system due to its readily available model library of sensitive components and its ability to accept third party models. Accurate modelling of mixed signal and digital circuits was not possible with this tool, but it did not affect the decision making process as alternative methods were used to capture their performance, and make sure their power consumption data was accounted for in the decision making.

The models of the EBW power source on the other hand are mixed-signal electronic circuits and were simulated in PSpice environment. Understanding and communicating the functionality of the existing design at a component level as well as at a system level was the main motivation of this work. The integration of the model formulated a complete system model of the EBW power source and its load that enabled assessing the system's ability on a fault handling mechanism for rapid weld recovery after flashover events; and analysis on power-stage output transient overshoots.

7.1 Discussion on LRUT pulser receiver model

In this work, all relevant subsystems in the pulser-receiver and its load were modelled in the LTSpice environment and characterised individually for validation, using a combination of theoretical calculations, measurement data and expert knowledge. Simulation covered the interaction that some of the subsystems can have when connected together with other subsystems so that any anomaly can be revealed. The first phase of this work was to compute power consumption data of the system to reliably specify the power source to the pulser-receiver. Having the ability of integrating and simulating the load model with the drive circuits for various load conditions and then using the knowledge gained from their collective performance to derive the specification for the drive circuits' power supply allowed the research to determine that the options for portability was overlooked and that the smaller size HV-Bank would jeopardise the system performance. The choice of full models used in most simulations allowed precise prototyping of hardware as well as facilitating new concepts to be tried.

Throughout all the case studies, a model evaluation performed against measurement data at component level was minimal. Instead, theoretical calculations, expert knowledge in the form of measurement data from previous hardware obtained in separate work and assumed subsystems' specifications were used. This was accepted as adequate, for two major reasons; first most models used in the development of all power sensitive sub circuits were topology specific models and were assumed accurate, second the equivalent circuit models developed for two main components – the battery and the transducer, which influenced the most in the decision making, were validated against measurement data. The acceptance of accuracy of equivalent circuit models of 20% and the attention paid in achieving agreement to measurement data only

within the application range satisfied the immediate research need, however this limits these models' ability for future adaption or enhancement of hardware.

While simultaneously simulating the complete system was encouraged at early stages in this research work to capture all inter-coupling nature between subsystems, it was not performed in the final model or in producing data to compute the overall power performance of the system. This simplification was considered acceptable, provided that the subsystems whose functionality affects the battery behaviour and the power consumption differently when connected to one or more subsystems are simulated together and accounted for in the power budget. This allows capturing change in power consumption due to the inter-coupling nature between constructs while help minimising complexity and improve simulation time.

Despite the model performance, in providing a platform, where various functionalities and design ideas can be assessed, extension of this work had been identified to make better use of this work in future. They are listed below as recommendations for further work:

1. Discrepancy between the measurement data and the simulation data of 20% for the transducer elements within the application range is acceptable. This is because; this discrepancy is also expected in reality between two different elements, because of the manufacturing procedures. However, agreement within 20% within the application range, though it satisfies the immediate need, limits the model's ability to support further enhancement of hardware. Moreover the model developed required refinements/ minor adjustments to the data produced by the approximated equations used to obtain model specific parameters. A review on this process to make the approximated equations, closer to reality would help advance the ability of the model.

2. It is beneficial if a platform/technology free template is provided for automating power budgeting purposes. In spite of the reliable computation of the power budget demonstrated, the individual simulation of model constructs and compiling the data separately is not ideal when assessment of a new configuration is carried out by a non-expert. Hence an adequate script or template with a built-in calculation-engine is needed, so that the power budget for this instrumentation can be automated for various configurations. A Microsoft Excel worksheet, built-in with a relevant engine is recommended to fulfil this. The data produced using simulation, can be formulated using curve-fitting methods, and the resultant equations can be used for building the engine behind the template.

7.2 Functional analysis of EBW power source

Derivation of optimum operating parameters for the enhanced EBW application was the main objective of this part of the research. This work introduced an application specific fault condition – flashover in an EBW application and critically analysed ways of enhancing the existing EBW power source hardware so that this fault condition can be handled better. Possibility of shortening the 30 ms weld dead-time, imposed by the fault handling circuits at detection of flashover to prevent damages to hardware and weld specimen, was investigated. Limiting conditions for the enhancement: the physical constraints imposed by the welding environment – Vacuum integrity with regards to gas behaviour were investigated; and hardware limitations likely to compromise the enhancement procedure were researched. The gap in the published research work on gas behaviour in a vacuum prevented making a decision on enhancing fault handling mechanism on the basis of the vacuum recovery time after flashover. Assessment of the limitations the hardware is likely to cause were researched using system

modelling techniques. All relevant subsystems in this hardware were modelled separately in a PSpice environment and characterised individually at a subsystem level for validation and to make them suitable for non-convergence free system level simulation. The validated subsystems were then integrated together to formalise the system level model and subsequent system level simulations.

The anticipated modification to a fault handling mechanism was expected to increase the stress on the hardware at a component level as well as a system level when the system responds to a flashover fault condition. For this reason a bottom-up design methodology was used in the system model development for achieving a near reality model. The multilevel abstractions of the system model and its ability to converge to cycle-by-cycle simulation made the research to perform a critical assessment of the system at a system level as well as a component level flexibly. Expert knowledge together with theoretical calculations and the measurement data obtained at a system level made it possible to evaluate the models at a system level. Component level measurements suffered due to a lack of resources and limited access, but did not make an impact in the decision making process due to the model accuracy maintained in this bottom-up development of the system model. Relaxation of resolution parameters in the simulation settings enormously helped achieve a non-convergent free simulation without compromising the quality of the simulation results. The quality of the data produced was assessed by comparing system model simulation results, against simulation data produced with a much tighter simulation setting during the characterisation of subsystem models.

The work not only analysed the EBW power source, but also delivered an accurate model for an off the shelf power device IGBT – DIM800DDM12-A000 to the knowledge base for public use (Parthipan et al. 2013). With the use of the developed system model, the research

revealed that the hardware could be modified to enhance the flashover fault handling mechanism by reducing the weld dead-time to 24 ms from 30 ms without compromising the hardware performance. But the consequence of this on weld quality and most importantly, whether this prediction agrees with real hardware tests are yet to be tested. A case study on the settling time of the power-stage output suggests there is room for improvement to optimise the weighting of the overshoot which occurs.

Implementation of the parameters derived in the real hardware and assessing the weld performance is recommended as further work.

REFERENCES

AboElFotoh, HMF, Iyengar, SS & Chakrabarty, K 2005, 'Computing reliablity and message delay for cooperative wireless distributed sensor networks subject to random failures', *IEEE transaction on reliablity*, 2005, pp. 145-155.

ADS803, TI 2002, *ADS803 datasheet*, viewed 14 Oct. 2012, <http://www.ti.com/lit/ds/symlink/ads803.pdf>.

Ali, E, Abdolhosein, N & Stoyan, BB 2005, 'Uninterruptable power supplies', in *Uninterruptable power supplies and active filters*, CRC Press, New York.

Aouzale, N, Chitnalah, A & Jakjoud, H 2009, 'PSpice modelling nonlinearity effects on ultrasonic waves', *International Conference on Microelectronics*.

Aouzale, N, Chitnalah, A, Jakjoud, H & Kourtiche, D 2007, 'PSpice modelling Diffraction Effects in Pulse Echo Ultrasonic System', *14th IEEE International Conference on Electronics, Circuits and Systems*.

Arash, R, Jafar, M, Kaveh, M & Mohammad, A 2008, 'Reliablity consideration for high power zero-voltage-switching flyback power supply', *IEEE 13th Internaltional power electronics and motion control conference*.

Arata, Y 1986, 'Electron and laser beam welding in Japan', *Electron and Laser Beam Welding*, Pergamon press, Tokyo.

Arata, Y & Tomie, M 1986, 'Development and Application of a Strong Focusing Electron Beam Gun With Mulit-Stage Electromagnetic Accelerating Units', in *Plasma, Electron and Laser Beam Technology*, American Society for Metals, Ohio, USA.

Athalye, G 1999, 'Design of pulse width modulation based smart switch controller for automotive applications', *IEEE 18th Digital Avionics System Conference*, 1999, pp. 8.C.1-1 - 8.C.1-7.

Avolution 2008, *Introducing ABACUS*, viewed 27 July 2013, <http://www.avolution.com.au/products>.

Azar, R, Udrea, F, Ng, WT, Dawson, F, Findlay, W, Waind, P & Amaratunga, G 2004, 'Advanced electro-thermal SPICE modelling of large power IGBTs', *IEE proc.-Circuits Devices Syst.*, June 2004.

Basso, CP 2008, 'Basic blocks and generic switched models', in P Scott (ed.), *Switch-mode Power Supplies*, McGrawHill, New York, USA.

Bello, V 1984, 'Using the SPICE2 CAD package for easy simulation of switching regulators in both continuous and discontinuous conduction modes', *Powercon*.

Benini, L, Castelli, G, Macii, A, Macii, E, Poncino, M & Scarsi, R 2000, 'A discrete-time battery model for high-level power estimation', *Proceedings of IEEE Europe conference and exhibition 2000 on design, automation and test*, 2000, pp. 35-39.

Benini, L, Castelli, G, Macii, A, Macii, E, Poncino, M & Scarsi, R 2000, 'A discrete-time battery model for high-level power estimation', *Proceedings of IEEE Europe Conference and Exhibition 2000 on Design*.

Binard, J & Ducrot, A 1986, 'Electron beam welding of thick materials with a 200kW gun', *1st conference on power beam technology*, Brighton.

Bluff, RJ 1998, 'Avionic system modelling', *Simulation '98. International Conference on*, vol 457, pp. 11-18.

Brodersen, AJ & Al-Ali, AR 1989, 'Macromodels for digital to analog converters', *IEEE Proceedings. Energy and Information Technologies in the Southeas*, vol 3, pp. 1235 - 1238.

BS EN 50324-1 2002, 'Piezoelectric properties of ceramic materials and components'.

BS EN ISO 14744-1 2008, 'British standard: BS EN ISO 14744-1', British standards.

Cadence - PSpice 2012, *Cadence PSpice A/D and Advanced Analysis*, viewed 26 July 2012, <http://www.cadence.com/products/orcad/pspice_simulation/pages/default.aspx>.

Cazes, R, Dard, P & Sayegh, G, 'Control of the electric discharge (flash) in electron beam welding gun'.

Chen, N, Deng, Y, Wu, J & He, X 2011, 'An efficient semi-mathematical model for co-pack IGBT', *Applied Power Electronics Conference and Exposition (APEC), 2011 Twenty-Sixth Annual IEEE*, pp. 1833 - 1837.

Christophe, PB 2008, 'Basic blocks and generic switched models', in *Switch-Mode Power Supplies Spice simulations and Practicaldesigns*, McGraw Hill, New York.

Christophe, PB 2008, 'Chapter 1: Introduction to power conversion', in *Switch-mode power supplies; Spice simulations and practical designs*, Mc Graw Hill, New York.

Cirrus Logic - Apex 2008, *PA90*, viewed 07 August 2012, <http://www.apexanalog.com.php53-23.ord1-1.websitetestlink.com/wp-content/uploads/2012/08/PA90U_N.pdf>.

Cirrus logic 2009, 'Application notes: Driving capacitive loads APEX-AN25', Application notes, Apex, Cirrus Logic, Cirrus Logic.

Committee, Working group C-5 of the systems protection subcommittee of the IEEE power system relaying 2000, 'Mathematical models for current, voltage and coupling capacitor voltage transformer', *IEEE transactions on Power Delivery*, Jan 2000, pp. 62 - 72.

Cotorogea, M 1998, 'Using analog behavioral modeling in PSpice for the implementation of subcircuit-models of power devices ', *Power Electronics Congress, 1998. CIEP 98. VI IEEE International*.

Donnelly, JM & Gauen, K 1993, 'New IGBTs and simulation models simplify automotive ignition system design', *Power Electronics Specialists Conference, 1993. PESC '93 Record., 24th Annual IEEE*, 1993, pp. 473-481.

Dynamic-ceramic_Dynallox96 2010, 'Datasheet: Material properties; Dynallox 96', Dynamic ceramic Ltd, Crewe.

Dynex semiconductor 2009, *DIM800DDM12-A000*, viewed 28 May 2012, <http://www.dynexsemi.com/assets/IGBT_Modules/Datasheets/DNX_DIM800DDM12-A000.pdf>.

Dynex semiconductor 2009, *DIM800DDM17-A000*, viewed 28 May 2012, <http://www.dynexsemi.com/assets/IGBT_Modules/Datasheets/DNX_DIM800DDM17-A000.pdf>.

EBL Piezoelectric precision 2010, *EBL Lead Zirconate Titanate Characteristics*, viewed 07 August 2012, <http://www.eblproducts.com/leadzirc.html>.

Fritz, D, Nazarenko, OK, Irie, H, Abe, N & Ohmine, M 1998, '40 years of electron beam welding and material processing', *6th International conference on welding and melting by electron and laser beams*, Toulon.

Gallagher, TJ & Pearmain, AJ 1983, 'Electrical insulating materials: Time lag to breakdown', in *High voltage measurement, Testing and Design*, John WIley & Sons, New York.

Gallagher, TJ & Pearmain, AJ 1983, *High Voltage Measurement, Testing and Design*, John Wiley & Sons Ltd, Bath, Great Britain.

Galsworthy, JC & Bird, J 1998, 'The electron beam welding of marine components', *6th International conference on welding and melting by electron and laser beams*, Toulon.

Gharaibeh, Y 2008, 'The Novel use of Long Range Ultrasonic Guided Waves in Complex Structures', EngD Report, Brunel University, London.

Gharaibeh, Y, Sanderson, R, Mudge, P, Ennaceur, C & Balachandran, W 2010, 'Investigation of the Behaviour of Selected Ultrasonic Guided Wave Modes to Inspect Rails for Long-Range Testing and Monitoring', *Journal of Rail and Rapid Transit*, vol 224, no. 3, pp. 1091 - 1104.

Gold, S 1997, 'A PSPICE Macromodel for Lithium-Ion Batteries', *12th Battery Conference on Applications and Advances*.

Gold, S 1997, 'A PSPICE Macromodel for Lithium-Ion Batteries', *12th Battery Conference on Applications and Advances*.

Golten, J & Verwer, A 1991, 'System time response', in *Control system design and simulation*, Mcgraw-Hill, London.

Haig, GA & Stavrou, P 2012, 'Defect detection for aircraft components: An approach using ultrasonic guided waves and neural networks', *NDT2012*.

Hanson, RC 1986, 'High energy beam (EB/laser) an overview comparison', *1st International conference on Power Beam Technologies*, Brighton.

Hefner, Jr, AR 1991, 'An investigation of the drive circuit requirements for the power insulated gate bipolar transistor (IGBT)', *IEEE Tranaction on power electronics*.

Hefner, Jr, AR 1995, 'Modeling buffer layer IGBT for circuit simulation', *IEEE Transaction on power electronics*.

Integrated Silicon Solution Inc 2012, *datasheet: IS61WV102416BLL CMOS RAM*, viewed 15 Oct. 2012, <http://www.issi.com/pdf/61WV102416ALL.pdf>.

Jiles D, C, Thoelke J, B & Devine M, K 1992, 'Numerical determination of hysteresis parameters for modeling of magnetic properties using the theory of ferromagnetic hysteresis', *IEEE transactions on Magnetics*, Jan 1992, pp. 27 - 34.

Jo da Silva, J, Wanzeller, G, de Almeida Farias, P & da Rocha Nato, JS 2008, 'Development of Circuits for Excitation and Reception in Ultrasonic Transducers for Generation of Guided Waves in Hollow Cylinders for Fouling Detection', *IEEE Transactions on Instrumentation and Measurement*.

Jo de Silva, J, Goncalves, WM, de, AFP & de, RNJS 2008, 'Development of Circuits for Excitation and Reception in Ultrasonic Transducers for Generation of Guided Waves in Hollow Cylinders for Fouling Detection', *IEEE transaction on instrumentation and measurement*, 2008, pp. 1149-1153.

Joakim, K 2002, 'The concept of IGBT modelling and the evaluation of the PSPICE IGBT model', MSc thesis, Alstom Power, Vaxjo.

John, HC, Andrei, V, Xiao-Chun, G, Peter, L & John, V 1991, 'Nonlinear transformer model for circuit simulation', *IEEE Transaction on computer-Aided Design*, 1991.

Johns, P 2012, 'HV transformer: 300kV'.

Jose, L, San, E & Antonio, R 2004, 'Models for piezoelectric transducers used in broadband ultrasonic applications', in *Piezoelectric Transducers and Applications*, Springer, New York.

Klemperer, O 1959, 'The electron mass and its relativistic change', in *Electron Physics - The physics of the free electron*, Butterworths Scientific publications, London.

Kraus, K, Turkes, P & Sigg, J 1998, 'Physics-based model of power semiconductor devices for the circuit simulator SPICE', *Power Electronics Specialists Conference, 1998. PESC 98 Record. 29th Annual IEEE*, May.

Kroeze, RC & Krein, PT 2008, 'Electrical Battery Model for Use in Dynamic Electric Vehicle Simulations', *IEEE Power electronics specialists conference*, 2008, pp. 1336 - 1342.

Kroeze, RC & Krein, PT 2008, 'Electrical Battery Model for Use in Dynamic Electric Vehicle Simulations', *IEEE Power Electronics Specialists Conference*.

Kuang, S, Barry, WW & Stephen, JF 2000, 'A review of IGBT models', vol 15, no. 6.

Lauwers, E & Gielen, G 1999, 'A power estimation model for high-speed CMOS A/D Converters', *Proceedings of Europe Conference and Exhibition on Design, Automation and Test*.

Lauwers, E & Gielen, G 2002, 'A power estimation models for analogue circuits for architectural exploration of integrated systems', *IEEE transactions on very large scale integration (VLSI) systems*, 2002, pp. 155-1162.

Lauwers, E & Gielen, G 2002, 'Power Estimation Models for Analogue Circuits for Architectural Exploration of Integrated Systems', *IEEE Transactions on Very Large Scale Integration (VLSI) Systems*, 2002, pp. 155 - 162.

Leach, MW 1994, 'Controlled-Source Analogous Circuits and SPICE Models for Piezoelectric Transducers', *IEEE transaction on ultrasonics, ferroelectrics, and frequency control*, Jan 1994.

Linear Technology - LTSpiceIV 2012, *Linear Technology - Design simulation and device models*, viewed 26 July 2012, <http://www.linear.com/designtools/software/#LTspice>.

LT3751 2008, *Datasheet*, viewed 23 September 2012, <http://cds.linear.com/docs/Datasheet/3751fc.pdf>.

LTSpiceIV 2009, *LTSpice user manual*, viewed 10 August 2013, <http://ecee.colorado.edu/~mathys/ecen1400/pdf/scad3.pdf>.

Luo, J & Jha, NK 2001, 'Battery-Aware Static Scheduling for Distributed Real-Time Embedded Systems', pp. 444 - 449.

Majed, AM Dec., 1994, 'Power supply for Electron beam welder', ERA Technology - Design division, Leatherhead.

Ma, CL & Lauritzen, PO 1991, 'A simple power diode model with forward and reverse recovery', *Power Electronics Specialists Conference, 1991. PESC '91 Record., 22nd Annual IEEE*, pp. 411 - 415.

Meleka, AH 1971, *Electron-Beam Welding: Principles and practice*, Mcgraw-Hill, Maidenhead.

Mentor Graphics 2012, *SystemVision - Virtual lab for mechatronics system design and analysis*, viewed 26 July 2012, <http://www.mentor.com/products/sm/system_integration_simulation_analysis/systemvision/>.

Metzner, D, Schafer, J & Chihao, X 2001, 'Multi domain behavioral models of smart-power ICs for design integration in automotive applications', *Behavioral Modeling and Simulation, 2001. BMAS 2001. Proceedings of the Fifth IEEE International Workshop on*, 2001, pp. 84 - 89.

Michael, VF 1998, 'Galileo-Probe Battery-Life Estimation', *Proceedings annual reliablity and maintainability symposium*, 1998.

Microsim Corporation 1997, *Circuit Analysis Software ; page 6 - 42*, viewed 21 August 2012, <http://perso.he-arc.ch/gerald.huguenin/PSPICE_users_guide_AD.pdf>.

Middlebrook, RD & Slobodan, C 1976, 'A general unified approach to modelling switching-converter power stages', *IEEE Power Electronics Specialists Conference*, Cleveland.

Min, C & Rincon-Mora, GA 2006, 'Accurate Electrical Battery Model Capable of Predicting Runtime and I-V Performance', *IEEE transaction on energy conversion*, 2006.

Mitsubishi Electric 1998, *Mitsubishi IGBT Modules CM1000HA-24H*, viewed 21 August 2012, <http://www.datasheetcatalog.org/datasheet/MitsubishiElectricCorporation/mXvszuw.pdf>.

Mitsubishi Electric 1998, *Using GIBT Modules*, viewed 21 August 2012.

Mizuno, M, Dairaku, M, Ohara, Y, Ozaki, A, Tanaka, S, Uede, T, Watanabe, K, Yamashita, Y & Yokoyama, K 1989, 'Inverter type high voltage DC power supply for negative-ion-based neutral beam injectors', *Fusion Engineering, 1989. Proceedings., IEEE Thirteenth Symposium on*.

Mohan, N, Robbins & Robbins, WP 1995, *Power Electronics: Converters, Applications, and Design*, John Wiley & Sons, New York.

Moreland, J 1998, 'Choosing a simulation Tool', *IEE Colloquium - Power Electronics and Simulation*, 1998.

Mudge, P 2006, 'Teletest specifications', Plant Integrity - TWI, Cambridge.

Mudge, P & Catton, P 2006, 'Monitoring of Engineering Assets using Ultrasonic Guided Waves', *Proceeding of the 9th European Conference on Non-Destructive Testing*, Berlin.

Mudge, P, Lank, A & Alleyne, D 1994, 'A long range method of detection of corrosion under insulation in process pipework', *The 5th European union hydrocarbons symposium*, Edinburgh.

Neal, B 2010, 'Production test data'.

Nightinggale, KR, Sanderson, A, Punshon, C & Werme, LO 1998, 'Advances in EB technology for the fabrication and sealing of large scale copper canisters for high level nuclear waste burial', *6th International conference on welding and melting by electron and laser beams*, Toulon.

O'neill, GD 1959, 'Electron tubes for low and medium frequencies', in *Radio frequency handbook*, Mcgraw Hill, New York.

Orcad Inc. 1999, 'Analog devices: IGBT', in *OrCad PSpice A/D Reference guide*, Orcad Inc., USA.

Panigrahi T, D, Panigrahi, D, Chiasserini, C, Dey, S, Rao, R, Raghunathan, A & Lahiri, K 2001, 'Battery life estimation of mobile embedded systems', *IEEE 14th international conference on VLSI design*, 2001, pp. 57 - 63.

Parthipan, T 2010, '24 month report: System modelling of ultrasonic non-destructive testing instrumentation', Brunel University, London.

Parthipan, T Sept, 2011, 'EngD porgress report: Electronic systems enhancement by system modelling - 36 month'.

Parthipan T C8117 2010, 'Design specification for C8117 - Custom made HT transformer for MK4 HTPSU', Billy Parthipan, Plant Integrity Ltd., Cambridge.

Parthipan, T, Nilavalan, R, Balachandran, W & Mudge, P 2011, 'Design and Analysis of an Ultrasonic NDT Instrumentation Through System Modelling', *International Journal for Modern Engineering*, 2011.

Parthipan, T, Ribton, C, Mudge, P, Rajagopal, N & Wamadeva, B 2013, 'Enhancement of High Voltage Electron Beam Welding Power Supply', *IEEE ISIE13 conference*, Taipai.

Pedram, M & Wu., Q 1999, 'Design Considerations for Battery-Powered Electronics', *Proceedings of the 36th Design Automation Conference*, New Orleans.

Peter, VS & P, AC 1976, 'Microdischarges and Microdischarge Simulation in Uniform-Field High-Voltage Vacuum Gaps', *IEEE Transaction on Electrical Insulation*.

Power integrations 2000, *Concept 2SB315A*, viewed 20 September 2012, <http://igbt-driver.com/products/scale-1-plug-and-play-drivers/2sb315a>.

Powerex Inc., *Using IGBT modules*, viewed 20 May 2012, <https://www.galco.com/techdoc/prx/cm100dy-24a_app2.pdf>.

PTV05010W 2005, *PTV0510W 8-A, 5-V INPUT NONISOLATED WIDE-OUTPUT ADJUST SIP MODULE*, viewed 13 Jan. 2013, <http://www.ti.com/lit/ds/symlink/ptv05010w.pdf>.

Punshon, CS, Sanderson, A & Belloni, A 1998, 'Reduced pressure electron beam welding for steel pipelines', *6th International conference on welding and melting by electron and laser beams*, Toulon.

Puttmer, A, Hauptmann, P, Lucklum, R, Krause, O & Henning, B 1997, 'SPICE Model for lossy piezoelectric transducers', vol 44, no. 1, pp. 60 - 66.

Quang, D, Deming, C & Martin, DFW 2010, 'Dynamic power estimation for deep submicron circuits with process variation', *Design Automation Conference (ASP-DAC) 15th Asia and South Pacific*.

Rakhmatov, DN 2005, 'Battery Voltage Prediction for Portable Systems', *IEEE International Symposium on Circuits and Systems*.

Rakhmatov, DN & Vrudhula, SBK 2001, 'An Analytical High-Level Battery Model for Use in Energy Management of Portable Electronic Systems', *IEEE/ACM international conference on computer aided design*, 2001, pp. 488 - 493.

Rakhmatov, DN & Vrudhula, SBK 2001, 'An Analytical High-Level Battery Model for Use in Energy Management of Portable Electronic Systems', *IEEE/ACM International Conference on Computer Aided Design*.

Ramos, A, San Emeterio, JL & Sans, PT 2000, 'Improvement in transient piezoelectric responses of NDE transceivers using selective damping and tuning networks', *IEEE transaction on Ultrasonics, Ferroelectrics and Frequency control*, 2000, pp. 826-835.

Ramy, A, Florin, U, Mahesh, DS, Gehan, A, Wai, TN & Francis, D 2004, 'Advanced SPICE modeling of large power IGBT modules', vol 40, no. 3.

Ramy, A, Florin, U, Wai, TN, William, F & Peter, W 2008, 'The current sharing optimization of paralleled IGBTs in a power module tile using PSpice frequency dependent impedance model', *IEEE Transaction on power electronics*.

Rao, R, Vrudhula, S & Rakhmatov, DN 2003, 'Battery modeling for energy aware system design', *Computer socity*, Dec 2003, pp. 77- 87.

Reinhold, G & Gleyvod, R 1975, 'Megawatt HV DC power supplies', *Nuclear science IEEE Transactions on*, 1975.

Reinhold, G & Minkner, R 1965, 'Stabilized DC Power Supplies for Beam Injection into Orbital Accelerators', *Nuclear Science, IEEE Transactions on*, vol 12, no. 3, pp. 820-823.

Russell, JD 1981, 'High Power Electron-beam Welding', *Aircraft Engineering and Aerospace Technology*, vol 53, no. 11, pp. 21-24.

Ryan, CK & Philip, TK 2008, 'Electrical Battery Model for Use in Dynamic Electric Vehicle Simulations', *IEEE Power electronics specialists conference*, 2008, pp. 1336 - 1342.

Sanderson, A 1978, 'Magnetic trap devices for the elemination of high voltage breakdown in electron-beam welding equipmwnt', *4th International conference on Advances in welding processes*, The welding institute, Harrogate.

Sanderson, A 1986, 'Recent innovations in high power electron beam equipment design for industrial welding applications', *International conferance on Power beam technology*, TWI, Brighton.

Sanderson, A & Ribton, CN 1998, 'The development of RF excited guns and intelligent power supplies for EBW at upto 150kW and 300kV', *6th International conference on wleding and melting by electron and laser beams*, Paris.

Sattar, A 2001, *Insulated Gate Bipolar Transistor (IGBT) Basics*, IXYS Corporation, viewed 27 August 2012, <http://www.ixys.com/Documents/AppNotes/IXYS_IGBT_Basic_I.pdf>.

Schulze, KR & Powers, DE 1998, 'Nonvacuum electron beam welding: Convenient and highly productive', *6th International conference on welding and melting by electron and laser beams*, Toulon.

Scott, M & Philip, M 2008, *Modelling and Experimental Validation of the Acoustic Electric Feedthrough Technique*, DTSO, Victoris, viewed 02 MARCH 2013, <http://www.dtic.mil/cgi-bin/GetTRDoc?AD=ADA496556>.

Shcherbakov, AV 2012, 'Switching processes in power sources for electron beam welding gas with formation of high-voltage breakdown', *Welding International*, vol 26, no. 3, pp. 221-226.

Sherrit, S, Leary, SP, Dolgin, BP & Bar-Cohen, Y 1999, 'Comparison of the Mason and KLM Equivalent Circuits for Piezoelectric Resonators in the Thickness Mode', *Proceedings of IEEE Ultrasonics Symposium*, 1999, pp. 921 - 926.

Shih, F-Y, Chen, Y-T, Chen, DY & Wu, Y-P 1995, 'A PSpice-compatible model of PWM IC for switching power converters with soft-start characteristic', *Power Electronics and Drive Systems, 1995., Proceedings of 1995 International Conference on*, Feb 1995, pp. 335-340.

Shilpa, S & Taranjit, SK 2003, *Convergence guide*, viewed 05 June 2013, <http://www.orcad.com/documents/community.an/pspice/PSpice_Convergence.pdf>.

Signal, C & Somanchi, S 1998, 'Automotive and aerospace circuit fault analysis', *17th IEEE Digital Avionics Systems Conference*, 1998, pp. 116/1-116/7.

Steele, J & Eddlemon, D 1993, 'Use high-voltage op amps to drive power MOSFETs', *Electronic design - Analog Applications*.

Svilainis, L & Motiejunas, G 2006, 'Power amplifier for ultrasonic transducer excitation', *Ultragarsas*.

Synopsys 2012, *Synopsys Saber platform*, viewed 26 July 2012, <http://www.analogy.com/Systems/Saber/Pages/default.aspx>.

Tang, Y 1998, 'MSc Thesis: High power inverter EMI characterization and improvement using auxiliary resonant snubber inverter', Virginia Politechnic Institute and State University, Blacksburg.

Thomas, SG, Linda, MT & Arthur, JB 1982, 'A sixteen-bit monolithic bipolar DAC', *IEEE Solids state circuits*, vol 17, pp. 1127 - 1132.

Uwe, D, Didier, C, Andreas, M & Jphann, WK, *Design Tools for Power Electronics: Trends and Innovations*, viewed 01 Apr. 2013, <http://www.pes.ee.ethz.ch/uploads/tx_ethpublications/drofenik_APE07_01.pdf>.

Vacuumschmelze 2012, *MEGAPERM 40L® Magnetic properties*, viewed 12 August 2012, <http://www.vacuumschmelze.com/index.php?id=334>.

Van Deventer, J, Lofqvist, T & Delsing, J 2000, 'PSpice simulation of ultrasonic systems', *Ultrasonics, Ferroelectrics and Frequency Control, IEEE Transactions on*, vol 47, no. 4, pp. 1014 - 1024.

Vinnikov, D & Laugis, J 2007, 'High voltage Auxiliary Power Supply with the simplified power circuit topology for the DC trains', *Electrical Power Quality and Utilisation, 2007. EPQU 2007. 9th International Conference on*.

Vladimirescu, A 1990, 'SPICE - The third decade', *IEEE 1990 Bipolar circuits and technology meeting 5.1*.

Vladimirescu, A 1994, 'Convergence advice', in *The spice book*, John Wiley & Sons, Inc., NewYork.

Vladimirescu, A 1994, 'Spice algorithms and options', in *The spice book*, John Wiley & Sons, Inc., New York.

Vladimirescu, A 1999, 'SPICE - The forth decade: Analog and mixed-signal simulation - A state of the art', *Semiconductor Conference, 1999. CAS '99 Proceedings. 1999 International*.

Wei, A 2007, 'The Expected Energy Efficiency Consumption of Wireless Distributed Sensor Networks Based on Node Random Failures', *Communications and Networking in China - CHINACOM*, China.

Xiaojin, Z, Hao, C, Yong, S & Meiyu, L 2007, 'Design of Piezoelectric Power Amplifier Used for Adaptive Structural Vibration Control', , *Proceedings of 8th International conference on Electronic Measurement and Instruments*.

Xilinx power estimator 2010, *Xilinx FPGA power avandtage at every level*, viewed 14 Oct 2012, <http://www.xilinx.com/products/technology/power/index.htm>.

Xilinx Spartan-3A DSP 2011, *datasheets*, viewed 14 Oct. 2012, <http://www.xilinx.com/support/documentation/data_sheets/ds705.pdf>.

APPENDIX A: MODELLING OF ELECTROMECHANICAL COMPONENTS IN SPICE

This appendix offers a quick review on the procedures followed to model the electromechanical components described in Chapters 3 and 4. The content of this appendix provides enough information to support the development of the electromechanical models used in the LRUT system: piezoelectric transducer element, LRUT transducer and the medium – the test specimen. Most of this work was based on the published work carried out by (Puttmer et al. 1997), (Scott & Philip 2008) and (Van Deventer, Lofqvist & Delsing 2000) on implementing the acoustic nature of piezoceramic materials using a lossy transmission line model (LTRA) in SPICE based simulators. A step-by-step procedure documented in the British standard for piezoelectric transducers (BS EN 50324-1 2002) for calculating a complete set of material coefficients of piezoceramics was also used in conjunction with the aforementioned literature to derive the models.

A 1. Lossy transmission line model

Lossy transmission line models are described using telegraphic equations. The time delay necessary in piezoceramics for a mechanical signal to travel from one side to the other, together with its acoustic attenuation can be represented using the transmission line models. The analogy of this is very well documented in (Van Deventer, Lofqvist & Delsing 2000). This model in SPICE can be described as below:

.model Name LTRA (len={length} R={resistance} L={inductance} C={capacitance})

The material data: cross-sectional area A, the density, mechanical quality factor Q_m, the speed of sound in the material V_p and characteristic frequency f_p can be used with Equation A- 1, Equation A- 2 and Equation A- 3 can be used to determine the L, C, R of the LTRA model to model an acoustic layer of the chosen material. An attenuation constant can also be utilised in the calculation if attenuation effects required in the modelling. This work did not include this as the attenuation effect is insignificant in the short length of material considered. The parameter len is the length (thickness) of the material. This model has been used extensively in this work to model all the concerned electromechanical components, discussed in the next section.

$$L \equiv A\rho \qquad \textbf{Equation A- 1}$$

$$C \equiv \frac{1}{A\rho V_p^2} \qquad \textbf{Equation A- 2}$$

$$R \equiv \frac{2\pi f_p L}{Q_m} \qquad \textbf{Equation A- 3}$$

A 2. Electromechanical model description

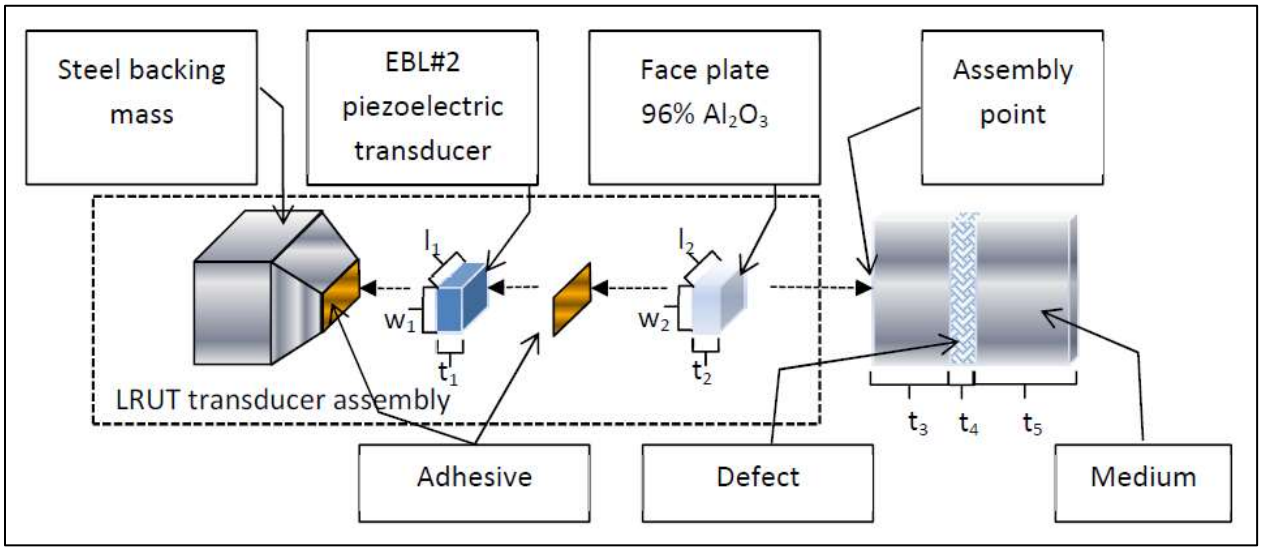


Figure A- 1 Schematic of the LRUT transducer and the test specimen

Figure A- 1 shows a schematic of the LRUT transducer and the medium (test specimen) that required modelling. The LRUT transducer is a hybrid of a steel backing mass, piezoelectric transducer of type EBL#2 (EBL Piezoelectric precision 2010) and a face plate of material Al_2O_3 (Dynamic-ceramic_Dynallox96 2010). An adhesive is used to glue the EBL#2 piezoelectric transducer to the steel backing mass and the face plate to the piezoelectric transducer. The medium is a metal piece that is under LRUT inspection. This case study used an aluminium plate of size 30 cm x 1 m x 10 mm. A defect of size 1 mm x 1 m x 10 mm was introduced in the middle of the medium for demonstration purposes.

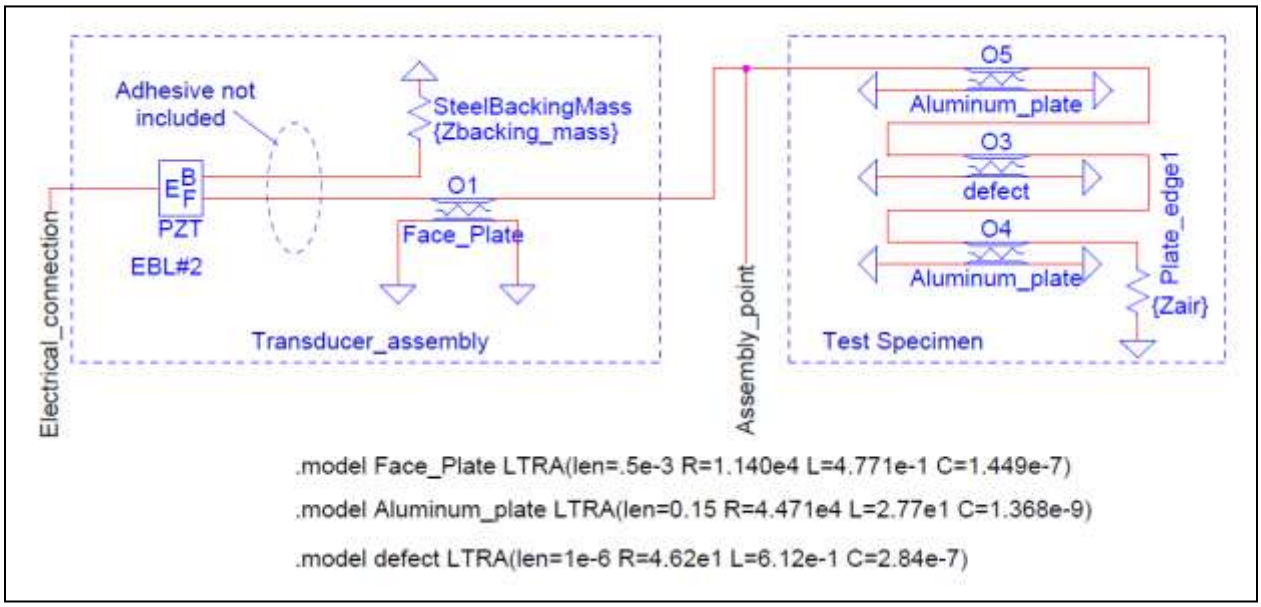


Figure A- 2 An equivalent circuit model developed in LTSpice

An equivalent circuit model of the schematic shown in Figure A- 1 is depicted in Figure A- 2. It consists of the equivalent circuit models of the piezoelectric transducer of type EBL#2, steel backing mass, face plate and the test specimen with a defect on it. The adhesive layers in the LRUT transducer assembly were not included, but can be included should the information on it be provided. The following sections describe the step by step procedure followed to model all

these components individually and integrating them to form the subsystem level components LRUT transducer and the medium that can be simulated in LTSpice environment.

A 3. Model development of EBL#2 piezoelectric transducer element

The LRUT transducer is built on a shear mode piezoelectric transducer EBL#2 (EBL Piezoelectric precision 2010) of dimension 13 mm (length - l) x 10 mm (width – w) x 0.5 mm (thickness – t). This research modelled this transducer element using the Puttmer model (Puttmer et al. 1997). Puttmer developed this model for a thickness mode piezoelectric transducer. But it was used to model this sheer mode transducer because, as (Jose, San & Antonio 2004), argues the thickness is very much smaller than the other two dimension and the effect on simulation would be insignificant. This model is included below in Figure A- 3 for completion.

```
.subckt pzt E B F
O1 B 1 F 1 mylossyTlineTrans
V1 1 2
E1 2 0 4 0 1
V2 E 3
C0 3 0 {c0T}
F1 0 3 V1 {hc0T}
F2 0 4 V2 {hT}
R1 4 0 1E3
C1 4 0 1
.model mylossyTlineTrans LTRA (LEN={thickT} R={RT} L={LT} C={CT})
.ends
```

Figure A- 3 Puttmer model (Scott & Philip 2008)

The model has three terminals: Electrical terminal E; Mechanical terminals: Back face B and front face F. The back face terminal B was used to add the backing mass model and the front

face terminal F was used to connect the face plate model. The model "mylossyTlineTrans" is a lossy transmission line model (LTSpiceIV 2009) that emulates the piezoceramic material properties of the EBL#2 transducer during simulation.

The functionality of the piezoelectric transducer is electromechanical and in the Puttmer model, the LTRA model represents the mechanical nature of the piezoceramic material. In the electrical section, the element C0 represents the static capacitance (mentioned and measured as free capacitance C^T in Chapter 3) of the component and it can be calculated using Equation A- 4 (Puttmer et al. 1997), (Van Deventer, Lofqvist & Delsing 2000). The clamped elastic modulus (C_{55}^D) can be extracted from the transducer's datasheet (EBL Piezoelectric precision 2010).

$$c0T = \frac{Clamped\ elastic\ modulus \times Area}{Thickness\ of\ the\ material} \qquad \textbf{Equation A- 4}$$

$$Gain_{F1} = h_t c0T \qquad \textbf{Equation A- 5}$$

$$h_t = \frac{Piezoelectric\ stress\ constant\ in\ the\ direction\ of\ propagation}{permitivity\ of\ free\ space} \qquad \textbf{Equation A- 6}$$

The interaction between the electrical and mechanical domain of the transducer was implemented using the current-control-current-source F1 and F2 in the Puttmer model as per its predecessor Leach model (Leach 1994). The gain factor for F1 was determined as a product of the transmitting constant h of the material and the static capacitance c0T as shown in Equation A- 5. Gain for F2 was h - the value of the transmitting constant. This value can be determined using Equation A- 6. The value of permittivity of free space is 8.8542e-12 $CV^{-1}m^{-1}$ (Scott & Philip 2008). Note that the suffix t implies a thickness mode operation and the same notation was kept as per the literature in this appendix to maintain uniformity.

- Determination of model parameters

The Puttmer model for EBL#2 piezoelectric transducer required six parameters: c0T, hc0T, hT, the RT, LT and CT to be inputted (specified within curly brackets) to make this model specific to EBL#2 piezoelectric element. A set of coefficients of the piezoceramics material properties of this transducer required determining, to calculate values for these parameters.

EBL#2 piezoelectric transducer data			
Parameters	**Symbol**	**Value**	**Unit**
Thickness	t	0.0005	m
Width	w	0.003	m
Length	l	0.013	m
Area	A	0.000039	m2
Density	ρ	7500	kg/m3
factor pi	π	3.141592654	
Free capacitance	C^T	1.1E-09	F
Permittivity of free space	e0	8.8542E-12	C/Vm
Series resonant frequency	fr	1650000	Hz
Parallel resonant frequency	fp	2350000	Hz
Mechanical quality factor	Qm	65	
Shear coupling factor	k_{15}	7.5	

Table A- 1 EBL#2 piezoelectric transducer data extracted from (EBL Piezoelectric precision 2010) and measurement

Table A- 1 lists the measured values: free capacitance C^T; characteristic frequencies f_r and f_p; and mechanical dimensions of an EBL#2 transducer and some of the extracted values from its datasheet (EBL Piezoelectric precision 2010). Table A- 2 lists the set of shear mode specific coefficients of piezoceramic properties that required determining. It also lists their corresponding approximate equations that allow determining these coefficients by plugging in the data provided in Table A- 1. The calculated values are the shear mode specific piezoceramic coefficients specific to EBL#2. British standard for piezoelectric properties of ceramic materials

and components (BS EN 50324-1 2002) documents these approximate equations as a step-by-step procedure for calculating a complete set of material coefficients of piezoceramics. Note that the piezoelectric coefficients determined were for a shear mode transducer. This is noticeable from the suffix (X_{55}) used on the coefficients.

Piezoelectric property	Description	Formula number	Formula	Calculated value	SI Unit
k_{15}	Sheer coupling factor	Eq 44/ datasheet	$\sqrt{((\pi f_r/2f_p)*\cot(\pi fr/2fp))}$	7.4654E-01	
S_{55}^D	Elastic compliance coefficient	Eq 18	$1/4*\rho*f_p^2t^2$	2.4144E-11	m^2/N
E_{11}^T	Piezoelectric stress constant	Eq 54	C^T*t/A	1.4103E-08	C/m^2
d15	Piezoelectric charge constant	Eq 49	$k_{15}*(E_{11}^T*S_{55}^E)$	4.3562E-10	C/N
S_{55}^E	Electric compliance coefficient	Eq 23	$S_{55}^D/(1-k^2_{15})$	5.4540E-11	m^2/N
E_{11}^S	Piezoelectric stress constant	Eq 56	$E_{11}^T(1-k_{15}^2)$	6.2429E-09	C/N
g_{15}	Piezoelectric voltage stress constant	Eq 50	d_{15}/E_{11}^T	3.0889E-02	Vm/N
C_{55}^D	Elastic stiffness constant	Eq 20a	$1/S_{55}^D$	4.1419E+10	N/m2
C_{55}^E	Elastic stiffness constant	Eq 20b	$1/S_{55}^E$	1.8335E+10	N/m2
V_5^D	sound velocity	Eq 36	$\sqrt{(C_{55}^D/\rho)}$	2.3500E+03	m/s

Table A- 2 Calculation of piezoelectric material properties of the EBL#2 transducer

Parameter	Description	Formula	Value	Effective Unit
c0T	Capacitance	E_{11}^S*e0*A/t	1.1000E-09	F
ht	Transmitting Constant	$d_{15}/(E_{11}^S*e0)$	1.6000E+09	Vm^2/CF
hc0t	Transmitting Constant	ht*c0T	1.8700E+00	FV/m
z0T	Acoustic Impedance	$\rho*V_5^D*A$	6.8738E+02	kg/s
LT	Lossy TN inductance	$z0T/V_5^D$	2.9250E-01	kg/m
cT	Lossy TN capacitance	$1/(V_5^D*z0T)$	6.1907E-07	s2/mkg
rT	Lossy TN resistance	$2\pi f_r*LT/Q_T$	6.6445E+04	kg/ms

Table A- 3 Model parameters for developing Puttmer model for EBL#2 transducer

Having determined the shear mode specific piezoceramic coefficients of EBL#2 all six parameters required by the Puttmer model to make it specific to EBL#2 can be calculated using equations A-1 to A-6. Table A- 3 summarise this and shows all six calculated values. Note the sound velocity (V_p) is referred to as V_5^D to match the approximation equation given by (BS EN

50324-1 2002). Equation numbers given in Table A- 2; column 3, corresponds to the equation numbers published by (BS EN 50324-1 2002).

The Puttmer model with the above parameters formed an equivalent circuit model for EBL#2. However, as the equations used were only 1st order approximations the simulation results produced by simulating this model did not produce results that closely matched the data provided by the transducer manufacturer EBL. Refinement of values $Gain_{F1}$ and $Gain_{F2}$ were required to make close mapping of results. The units given in Table A- 3 are effective units derived from the dimensions of each parameter.

A 4. Modelling of backing mass

The backing mass used in the assembly is a steel block of dimension 13 mm x 10 mm x 10 mm. In the equivalent circuit described in Figure A- 2, it was represented using a simple resistor of value $R_{SteelBackingMass}$. This value was calculated using Equation A- 7 (Leach 1994). The terms used in the equation and their values are presented in Table A- 4. For this simulation a 6 kΩ resistor was used to represent a steel block.

$$R_{acoustic} = \rho \times A \times V_p$$

Equation A- 7

Steel block of dimension 13 mm x 10 mm x 10 mm				
Symbol	**Description**	**Equation** (BS EN 50324-1 2002)	**Value**	**Unit**
ρ	Density		7.850E+03	kg/m^3
C_{55}^D	Modulus of elasticity		2.690E+13	Nm^2/kg
V_p	Sound velocity	$sqrt(C_{55}^D/\rho)$	5.854E+04	m/s
A	Area	length x width	1.300E+02	mm^2

Table A- 4 Material properties used to calculate acoustic impedance of the Steel backing mass

A 5. Modelling of face plate

The face plate was also modelled using a lossy transmission line model LTRA. Its model description is given in Figure A- 2 as a face plate. The characteristic parameters that required determining (R, L, C) to make the LTRA model specific to an Al_2O_3 face plate of dimensions 13 mm x 10 mm x 0.5 mm the procedure presented in Table A- 5 was followed. The determined values for LT, CT and RT tabulated in Table A- 5; column 4, were plugged in the face plate model during simulation. Validation of this model was not possible due to unavailability of an electrical terminal on the real face plate for impedance analysis.

Face plate of dimension 13 mm x 10 mm x 0.5 mm				
Symbol	**Description**	**Equation** (Scott & Philip 2008)	**Value**	**Effective Unit**
ρ	Density		3.670E+03	kg/m^3
C_{55}^{DT}	Modulus of elasticity		5.310E+10	Nm^2/kg
QT	Mechanical q factor		1.000E+03	
V_5^D	Sound velocity	$sqrt(C_{55}^D/\rho)$	3.804E+03	m/s
fp	parallel resonant frequency	$V_{55}^D/2*t$	3.804E+06	Hz
z0T	Acoustic impedance*area	$\rho V_{55}^D A$	1.815E+03	kg/s
LT	Lossy transmission line inductance	$z0T/V_{55}^D$	4.771E-01	kg/m
CT	Lossy transmission line capacitance	$1/z0T*V_{55}^D$	1.449E-07	s2/mkg
RT	Lossy transmission line resistance	$2*\pi*fp*LT/QT$	1.140E+04	kg/ms

Table A- 5 Face plate model parameters calculation, using Al_2O_3 material properties

A 6. Medium – the test specimen

The medium was also modelled using the lossy transmission line model LTRA. In Chapter 4, the case study used an aluminium plate of dimensions 0.3 m x 1 m x 10 cm as an example, where a defect of dimensions 1mm x 1m x 10 cm was introduced. The defect was also

modelled using the lossy transmission line model. The values of the parameters used in the model are presented in Table A- 6. It also includes the procedure for determining these values.

Aluminium plate of dimension 0.3 m x 1 m x 0.01 mm				
Symbol	**Description**	**Equation** (Scott & Philip 2008)	**Value**	**Unit**
ρ	Density		2.770E+03	kg/m^3
C_{55}^D	Modulus of elasticity		7.310E+10	Nm^2/kg
QT	Mechanical q factor		1.000E+03	
V_5^D	Sound velocity	$sqrt(C_{55}^D/\rho)$	5.137E+03	m/s
fp	parallel resonant frequency	$V_{55}^D/2*t$	2.569E+05	Hz
z0T	Acoustic impedance*area	$\rho V_{55}^D A$	1.423E+05	kg/s
LT	Lossy transmission line inductance	$z0T/V_{55}^D$	2.770E+01	kg/m
CT	Lossy transmission line capacitance	$1/z0T*V_{55}^D$	1.368E-09	s2/mkg
RT	Lossy transmission line resistance	2*π*fp*LT/QT	4.471E+04	kg/ms

Table A- 6 Material properties of the medium and its calculated model specific values

The test specimen had a defect introduced in the middle of it. Hence the equivalent circuit model was constructed using two separate lossy transmission line models with a third lossy transmission line model in between them to represent the defect. The model parameters for the defect model is also included in Figure A- 2, but the procedure is not described in this appendix as it was similar to the procedure followed for the face plate and medium.

A 7. Integration of models

The models of the piezoelectric transducer, backing mass, face plate and the medium were integrated together to form an equivalent circuit model of the LRUT transducer and the medium. This model can be simulated by connecting relevant models of electronic circuits as discussed in Chapter 4.

APPENDIX B: ELECTRON BEAM WELDING POWER SOURCE MODELS

The circuits and component models discussed in Chapters 5 and 6 are provided in this appendix with a brief introduction to them. All sub circuits other than the power stage (inverter and the high voltage transformer circuit) and the EBW-Gun were modelled using topology specific models, meaning all of their models are a direct translation of their schematics. The power stage and the EBW-Gun were modelled using their corresponding equivalent circuit models.

The hardware can be configured with different values of components to change their functionality. Three such functions this research looked at were the kV (acceleration voltage) recovery time after a flashover fault condition, overcurrent detection in the inverter due to fault conditions at the load and the power stage output overshoot damping that was discussed in Chapter 6. This appendix provides the values of the components that set different rates at which kV recovers after flashover and the values of the component that set different levels of the overcurrent threshold limit. The procedure for altering the weighting of the power stage overshoot is discussed in detail in section 6.7.1. The traces showing the effect of changing this weighting are also presented.

All auxiliary supplies that power up the circuits were modelled using a voltage source model (VDC) provided in the PSpice: source library and all sensitive nodes and some ideal interconnections between components or subsystems that triggered non-convergence issues were introduced with small values of resistors and/ or inductors. This is noticeable in most of the circuit models included in this appendix.

B 1. Power stage – The H-Bridge inverter

The inverter was constructed of four IGBTs of manufacturer's part number DIM800DDM12-A000 (Dynex semiconductor 2009). This part is a Co-Pack. The PSpice circuit simulation model developed for the inverter is shown in Figure B- 1.

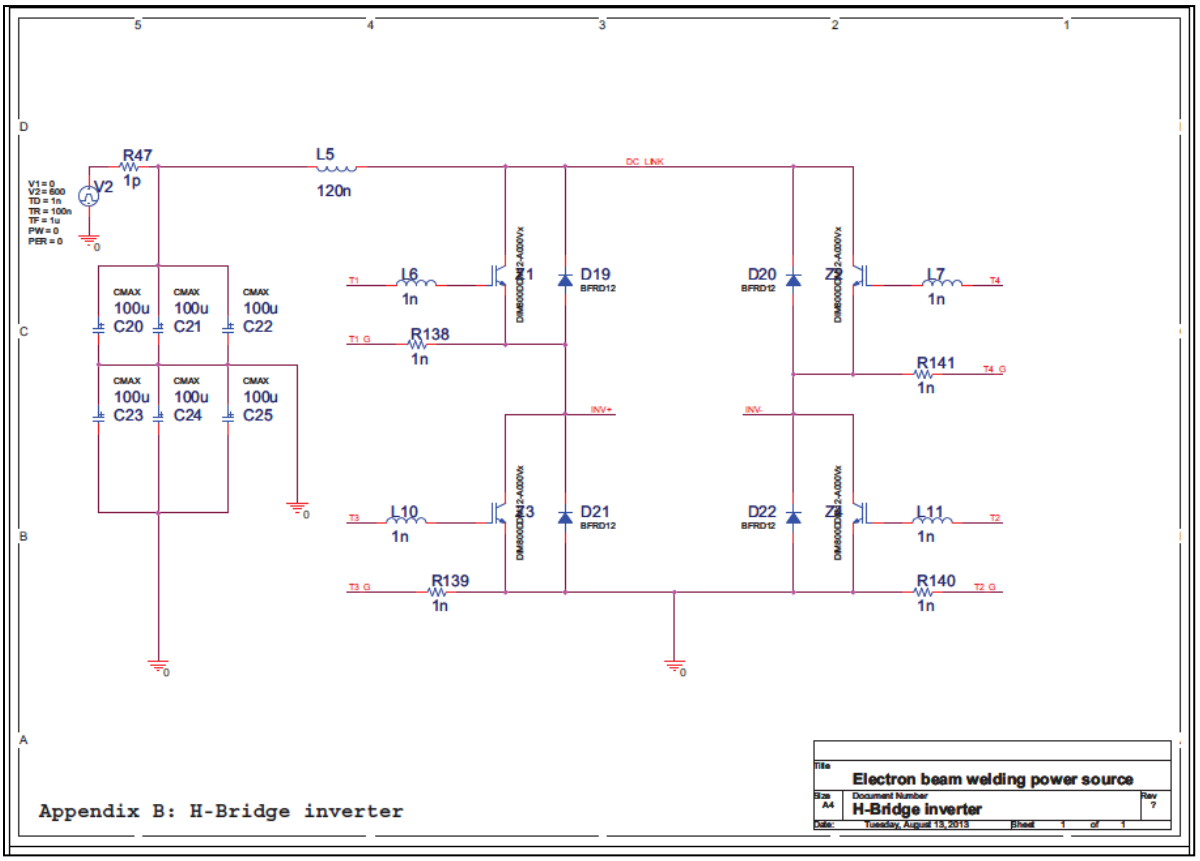


Figure B- 1 H-Bridge inverter model based on IGBT DIM800DDM12-A000 model

It includes the equivalent circuit models of the IGBTs, free wheel diodes, bulk capacitor bank and a voltage source. The models of the capacitor bank and the voltage source V2 were modelled using ideal PSpice models of capacitors in the PSpice: analog library and a pulse voltage source (VPULSE) in the PSpice: source library respectively. The initial value of the V2 was set to 0 V and was programmed to ramp up in 10 ns to its corresponding value (600 V_{DC}) 10

ns after a simulation time t = 0 seconds. This helped achieve a non-convergence free simulation. Other models used in the model development were developed as per their corresponding real counterparts. In the real hardware the –Ve bus bar is isolated from the GND. But in this simulation model this node was connected to the GND to provide a reference point. This helps PSpice to work out the DC operating points which is crucial for transient simulations. This does not affect the design principle of the inverter in simulation.

a. IGBT model

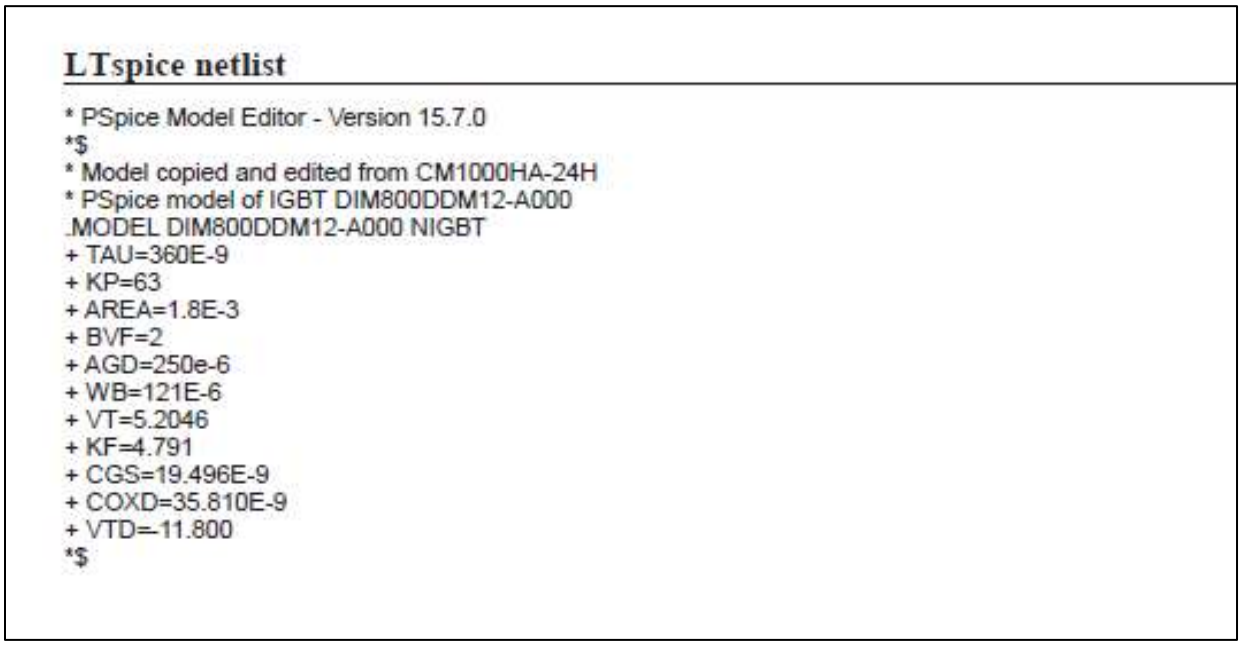


Figure B- 2 PSpice model for DIM800DDM12-A000

The PSpice model developed for the IGBT of manufacturer's part number DIM800DDM12-A000 is depicted in Figure B- 2. This model is based on a Hefner model (Hefner, Jr 1995). The model parameter description and the values inputted in the model are listed in Table B- 1. The free wheel diode of this device was modelled separately as shown in Figure B- 3.

Parameter	Description	DIM800DDM1200-A000	Unit
TAU	Ambipolar recombination lifetime	3.60E-07	s
KP	Internal MOSFET transconductance	63	A/V^2
AREA	Active area of the device	1.80E-03	m^2
BVF	Avalanche uniformity factor	2	
AGD	Gate-Drain overlap area	2.50E-04	m^2
WB	Metallurgical base width	1.21E-04	m
VT	Threshold voltage	5.20E+00	V
KF	Internal MOSFET linear region transconductance	4.79E+00	A/V^2
CGS	Internal MOSFET gate-source capacitance per unit area	1.95E-08	F/cm^2
COXD	Internal MOSFET gate-drain overlap oxide capacitance per unit area	3.58E-08	F/cm2
VTD	Internal MOSFET gate-drain overlap depletion threshold	-1.18E+01	V

Table B- 1 Model parameters for DIM800DDM12-A000 PSpice model

LTspice netlist

```
* FWD Diode model based on the PSpice default power diode
*$
.MODEL DIM800DDM1200_A000_FWD D
+ IS=.1
+ N=4.6905
+ RS=780.22E-6
+ IKF=13.327
+ CJO=1.0000E-12
+ M=.3333
+ VJ=.75
+ ISR=1.0000E-12
+ BV=1.1995E3
+ IBV=25.866E-6
+ TT=431.48E-9
*$
```

Figure B- 3 PSpice model for the free wheel diode of DIM800DDM12-A000

B 2. Power stage - High voltage transformer and the EBW-Gun

This circuit is the high voltage conversion stage where the output of the inverter (single phase ±600V at 5 kHz) is converted to 330 kV DC voltage. The circuit model developed for this subsystem is shown in Figure B- 4. It consists of a number of component models and a brief description of the main component models are given below:

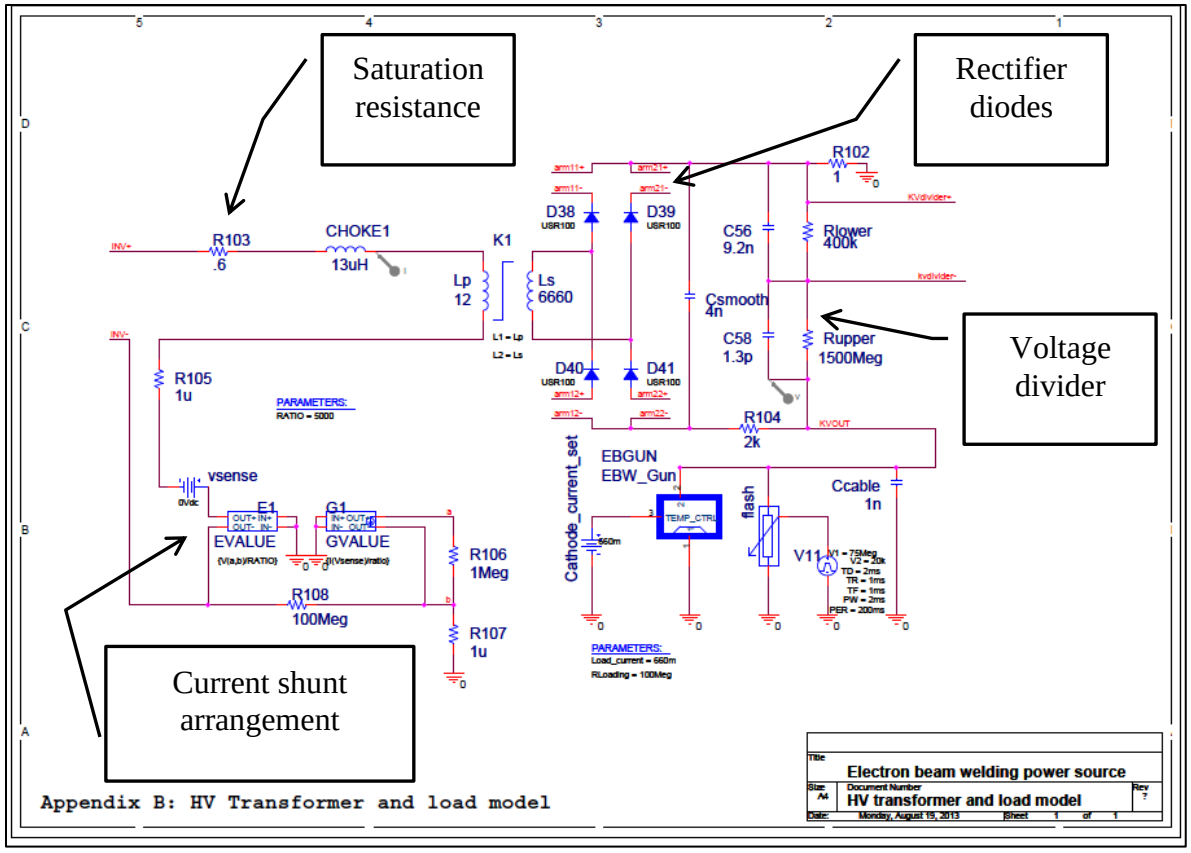


Figure B- 4 Models of high voltage transformer arrangement and load

a. Choke

The choke (labelled as choke1) is an air core inductor model. It was modelled using an ideal inductor.

b. High voltage transformer

The high voltage (HV) transformer was modelled using the Jiles-Atherton model (Jiles D, Thoelke J & Devine M 1992). The core details were calculated based on the information provided by the transformer manufacturer (Johns 2012). Parameters inputted in the model are presented in Table B- 2. The saturation current when the HV transformer saturates was set by the resistor value R103.

Parameter Name	Description	Value
LENGTH	Core path length	224 cm
AREA	Cross sectional area	22 cm^2
GAP	Core gap	0.26e-6
BR	Residual flux density	7500 Gauss
BM	Saturated flux density	14800 Gauss
HC	Coercive magnetic force	0.0753982 Oersted

Table B- 2 HV Transformer core parameters included in the Jiles-Atherton model

c. High voltage rectifier

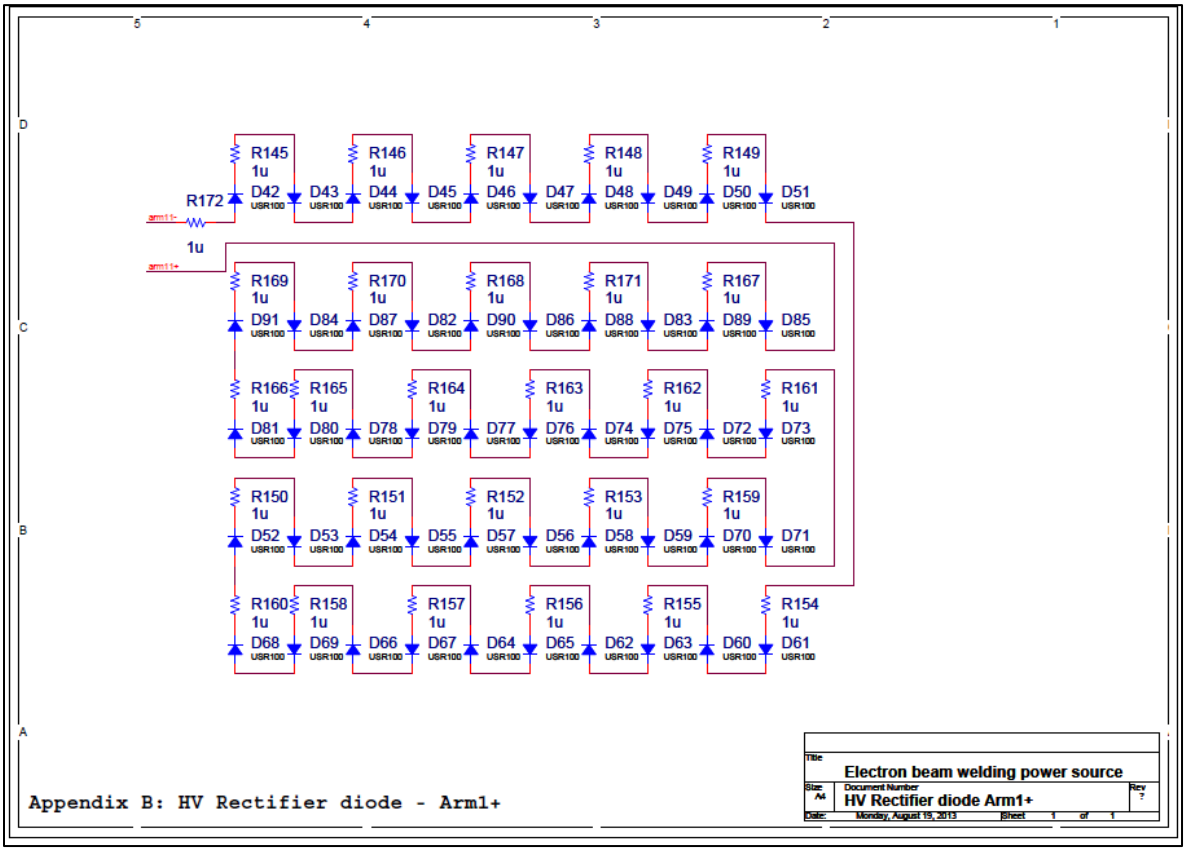


Figure B- 5 High voltage rectifier diode arrangement

The high voltage rectifier was fabricated by stacking a number of high voltage diodes of manufacturer's part number USR100.It is a 10 kV diode. Its model can be found in the PSpice: DIODE HV library. These are 10 kV diodes. The circuit model developed for the stack of diodes connected to each arm of the rectifier is shown in Figure B- 5.

d. High voltage resister network – Voltage divider

The high voltage resister network (R_{lower} and R_{upper}) together with the circuit components built in the kV demand circuit (B 4: b) forms a voltage divider to produce a 40 V DC full-scale voltage feedback signal when the power stage output is 300 kV, i.e., the scale was set as 300 kV = 40 V (7500:1). The capacitor divider network connected parallel to this resistor network also has the same ratio, providing compensation for the unintentionally introduced capacitance inherited from the high voltage resistors R_{lower} and R_{upper} by the transformer arrangement.

e. Load – EBW gun

LTspice netlist

```
* PSpice Model Editor - Version 15.7.0
*$
.SUBCKT EBGUN 1 2 TEMP_CTRL
R1 1 2 1E10
G1 1 2 Value = {(0+((V(TEMP_CTRL))/(pwr(((1+7e-9*exp(-1e-3*(V(1,2)-60k)))),100e-3)+1u)))}
.ends
*$
```

Figure B- 6 EBW Gun model

The EBW-gun behaviour was modelled using an S-curve equation as shown in Figure B-6. This model sets the leakage current of the EBW-Gun to 1 mA and the threshold voltage of 60 kV. The temperature control terminal TEMP_CTRL in the model was used to adjust the gain value of the G-Value (G1) to imitate the thermionic effect of the cathode shown in Figure 6-14.

B 3. IGBT drive circuits

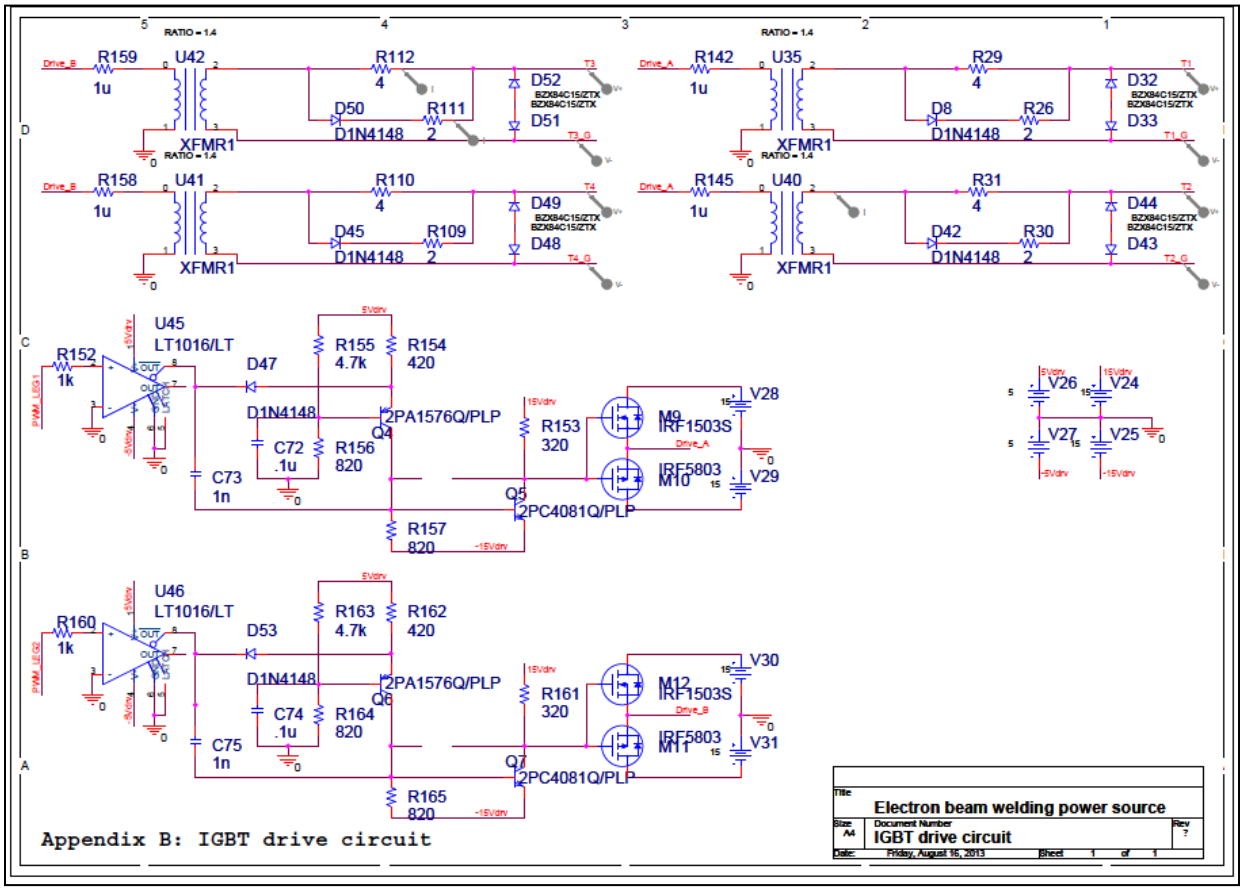


Figure B- 7 IGBT drive circuits

The gate drive circuit (Power integrations 2000) used in the real hardware was not modelled because the schematics of its circuits were not available. The modelled circuit for the performing gate drive functionality of this system is shown in Figure B- 7. This circuit matched the following aspects of the real circuit:

- The gate resistor arrangement was formed using a resistor of value 4 Ω in parallel with a serially connected IN4148 diode and a resistor of value 2Ω (e.g., R112 is connected in parallel to the serially connected D60 and R111). This was to provide different turn-on and turn-off time for the IGBTs.

- Back to back protection diodes were used to emulate protection of the gate of the IGBTs against gate drive over voltage signals (e.g., Zener diodes D51 and D52 of manufacturer's part number BZX84C15). This limits the switching signal level to +/- 15V maximum.
- 5V logic unipolar PWM output signals (PWM_LEG1 and PWM_LEG2) were level shifted to +/-15V logic. This functionality was facilitated by the circuit associated with U45 and U46.

All these features were considered as the main aspects of the IGBT drive circuit required for matching the real hardware. The functional performance of this circuit model switching two opposite IGBTs (Figure B- 1:Z3 and Figure B- 1:Z4) is presented in Figure B- 8. The traces included in it are listed below:

- B1: Gate current during turn on and turn off. A difference in the rate of change of gate charge and discharge current are noticeable.
- B2: Switching signals across the gate and emitter of the IGBTs (V_{GE}). Turn-on voltage and turn-off voltages are +15V and -15V respectively. The minimum dead time of 20 % between the switching of Z3 and Z4 is also noticeable. This was set by the parameter dead-time in the PWM model discussed in section 6.3. In the real hardware it is set by the hardware.
- M1: inverter current switches through Z3, showing approximately 900 A switching through the device.
- M2: The V_{CE} across Z3 when the inverter current of value 900 A is switched through Z3

- T1: inverter current switch through Z4, showing approximately 900 A switching through the device.
- T2: The V_{CE} across Z4 when the inverter current of value 900 A is switched through Z4

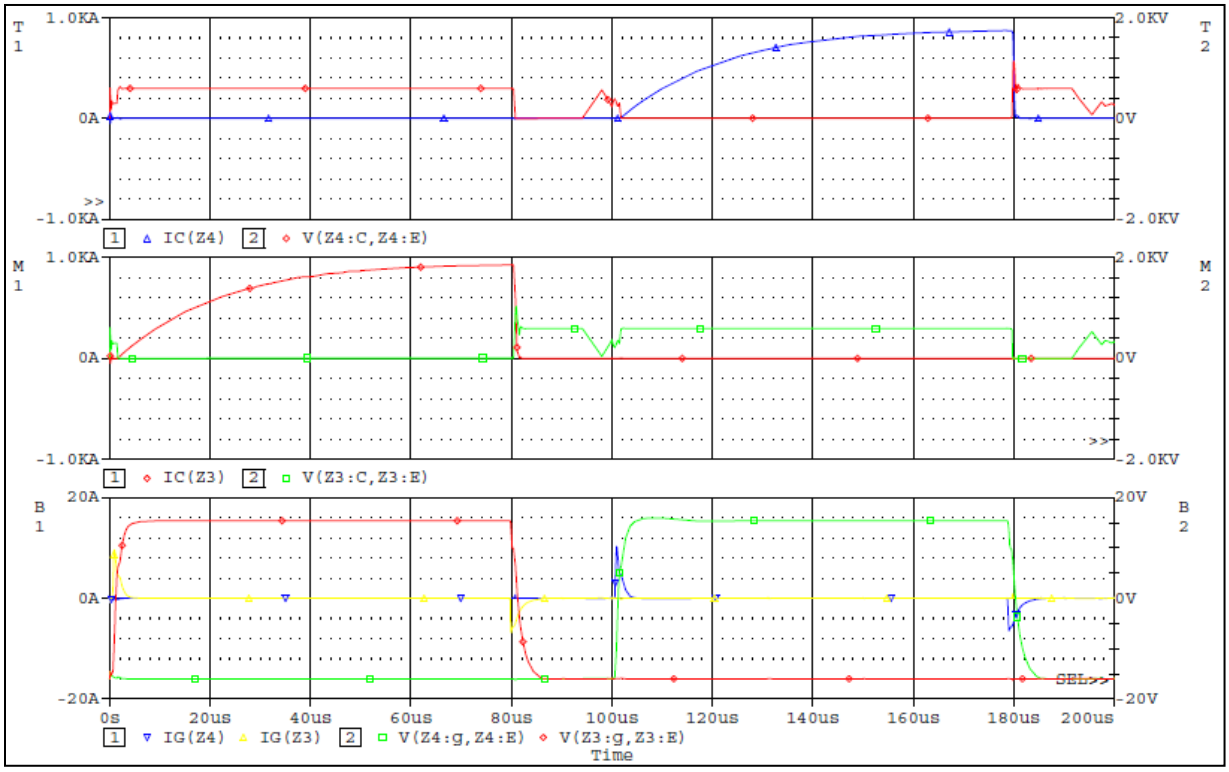


Figure B- 8 IGBT driver circuit functionality of driving IGBTs Z3 and Z4

B 4. Control circuits

The inverter control circuit design mainly includes generation of PWM switching signals, inverter current monitoring and fault detection circuits necessary for safe operation. The schematics of the real hardware that perform these functions were translated into topology specific models. The functionalities implemented in the control circuit model are discussed below:

a. PWM Circuit

The circuit model generates a natural PWM with a built in dead time of 20%. Figure B-9 is the PSpice circuit model developed to emulate the PWM functionality of the system. The switching frequency was set to 5 kHz. The feedback loop for the compensation circuit discussed in section 6.3 was modelled using C69 and R143. The PWM model generates an internal clock and the ramp signals and sets the dead time.

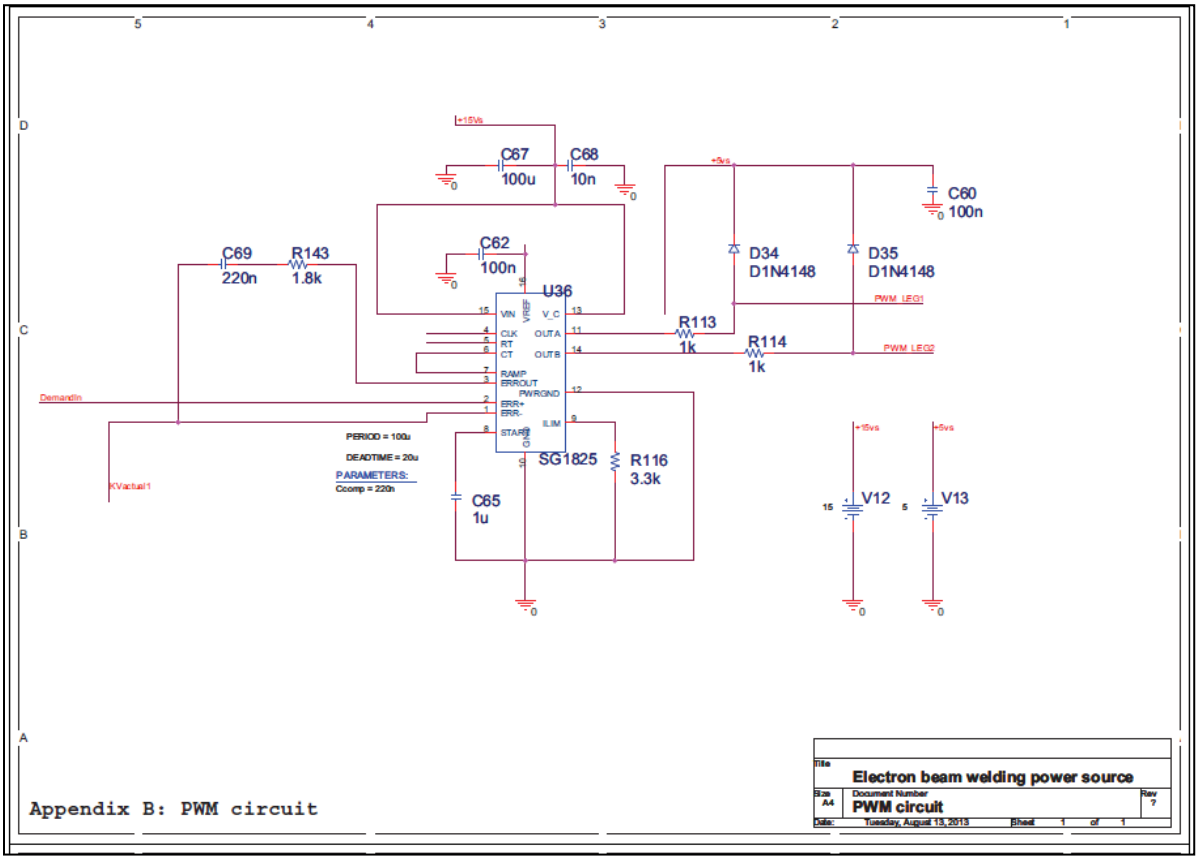


Figure B- 9 Control circuit model: PWM circuits

b. kV demand circuit

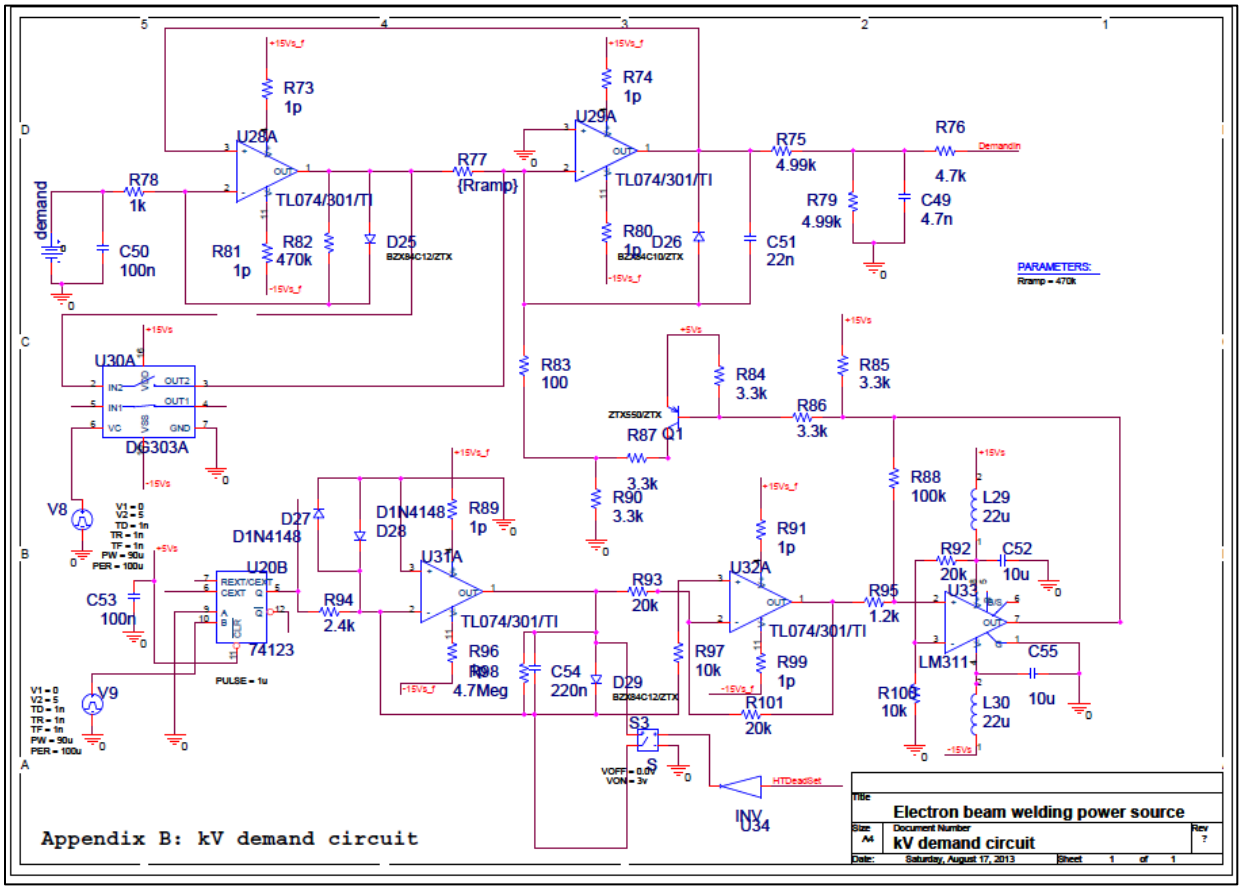


Figure B- 10 Control circuit model: a kV demand circuit

This circuit produces a kV demand signal. This circuit also includes features that govern the weld dead time discussed in section 6.7.2. Its schematic was translated into a topology specific circuit model as shown in Figure B- 9.

The shortening of the weld dead time discussed in section 6.7.2 is composed of two parts: the dead time and the ramp time as shown in Figure B- 11.

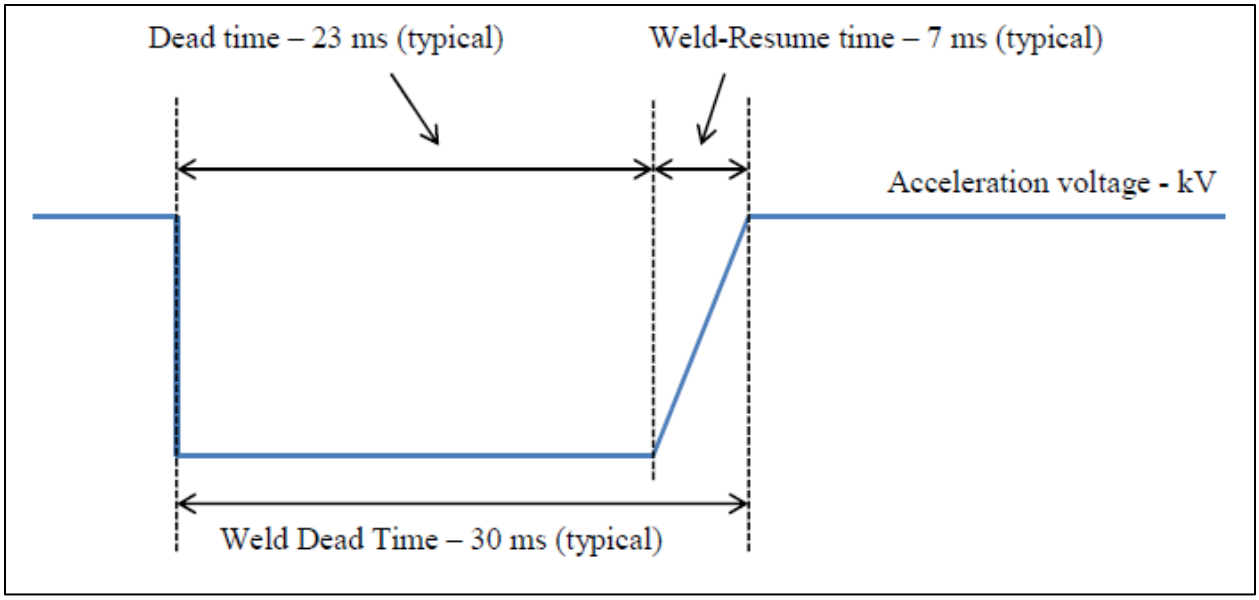


Figure B- 11 Weld dead time

The modelling of this aspect is discussed below:

- Dead time: This parameter is set to approximately 23 ms in the real hardware. This is facilitated by the circuit formed by components U20B, U31A, U32A, U33, U34, S3 and Q1. The signal HTDEADSET is set when a flashover is detected by the fault detection circuit discussed in section B 4: 0. When this signal is set, the output of U31A is pulled down to 0 V. This action subsequently sets the signal $V_{demandin}$ to 0 V by turning on the clamping transistor Q1, which consequently initialises the dead time. This dead time is reset by the integrating circuit formed by the operational amplifier circuit formed by U31A. The charging time constant of this circuit is 528 µs (R94*C54). The U20B - retriggerable monostable multivibrator model of 74HC123, produces a train of pulses of pulse width 1 µs at a frequency 10 kHz (period 100 µs). These pulses charge the capacitor C54 to the maximum of 12V (limited by D29), at a rate of 22.7 mV per pulse. This integrating action subsequently turns off the clamping

transistor Q1, when the voltage across the capacitor C54 - V_{C54} rises above the set threshold voltage of the comparator circuit formed by the simulation model U33 of the voltage comparator LM311. In the existing hardware this threshold value is set to 5V. This implies the charging process requires just over 220 of the aforementioned pulse to reach this threshold value (5V/22.7 mV). This equates to approximately 23 ms (222.26 cycles x 100 µs = 22.26 ms to be exact). A simulation illustrating this is shown in Figure B- 12.

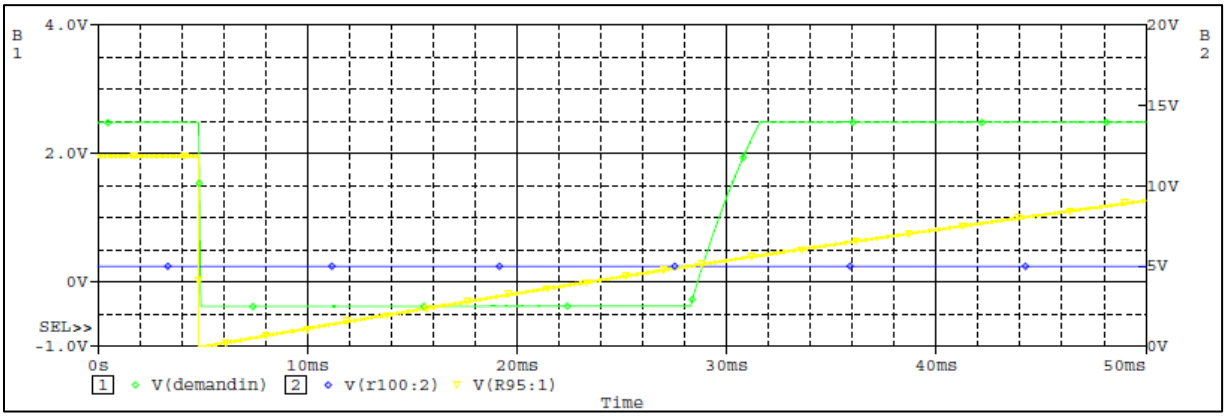


Figure B- 12 Simulation showing 23 ms dead time and 7 ms ramp time

- The ramp time: In the real hardware this value is set to 7 ms. This parameter is set by the components C51 and R_{ramp}. The ramp time can be changed by adjusting the value of the resister R_{ramp}. The values of R_{ramp} for setting different ramp times are given in Table B- 3.

Ramp time	Value od C51	Resistor value
7 ms	22 nF	470 kΩ
6 ms	22 nF	395 kΩ
5 ms	22 nF	330 kΩ
4 ms	22 nF	264 kΩ
3 ms	22 nF	198 kΩ
2 ms	22 nF	132 kΩ
1 ms	22 nF	65.9 kΩ
500 us	22 nF	32.9 kΩ

Table B- 3 Ramp time setup

c. kV actual circuit

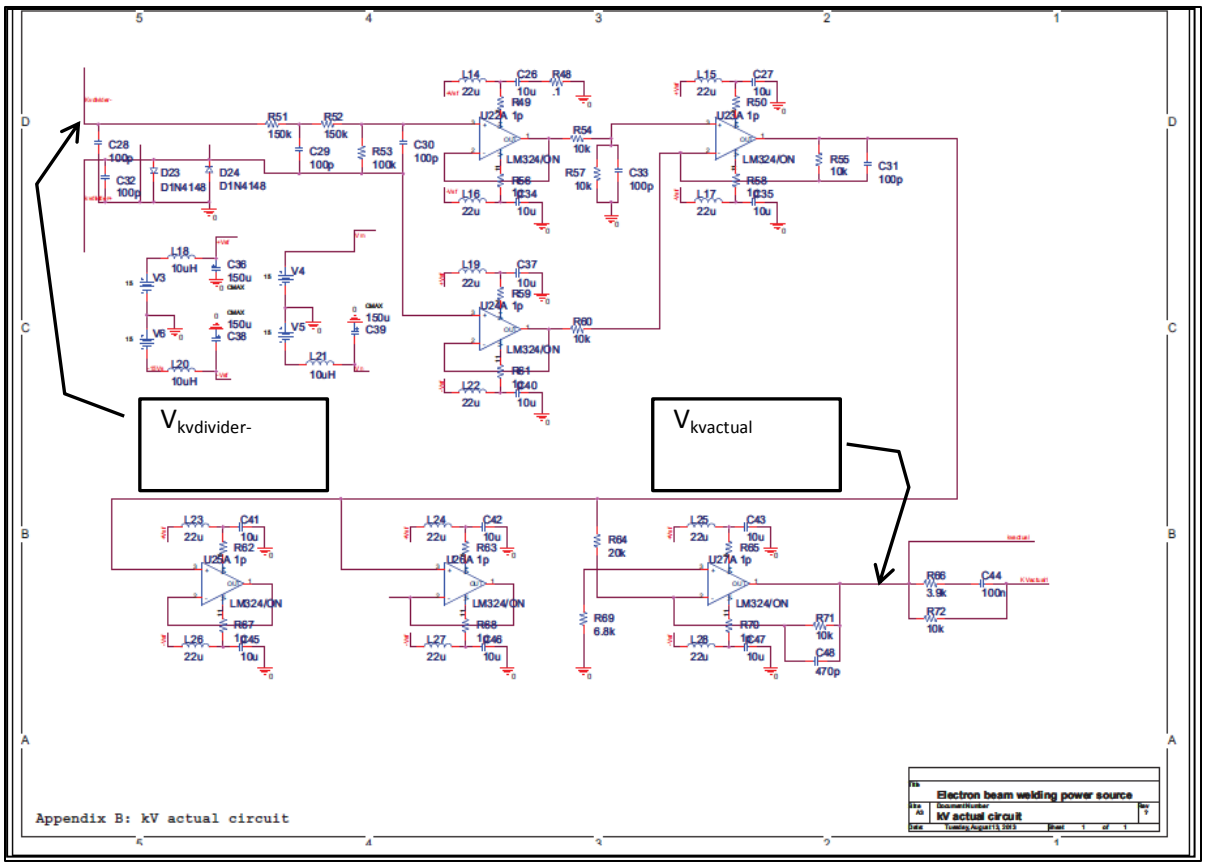


Figure B- 13 Control circuit model: kV actual circuit

This circuit signal condition the voltage feedback signal generated at the output of the power stage by the voltage divider network discussed in B 2: d. Its topology specific circuit model is shown in Figure B- 13. The output of this circuit model provides a 5V full scale output signal $V_{kvactual1}$ to represent the acceleration voltage of a value 300 kV. This output signal is fed into the error amplifier built in the PWM model for comparison with the acceleration voltage demand signal $V_{DemandIn}$.

d. Inverter current measurement

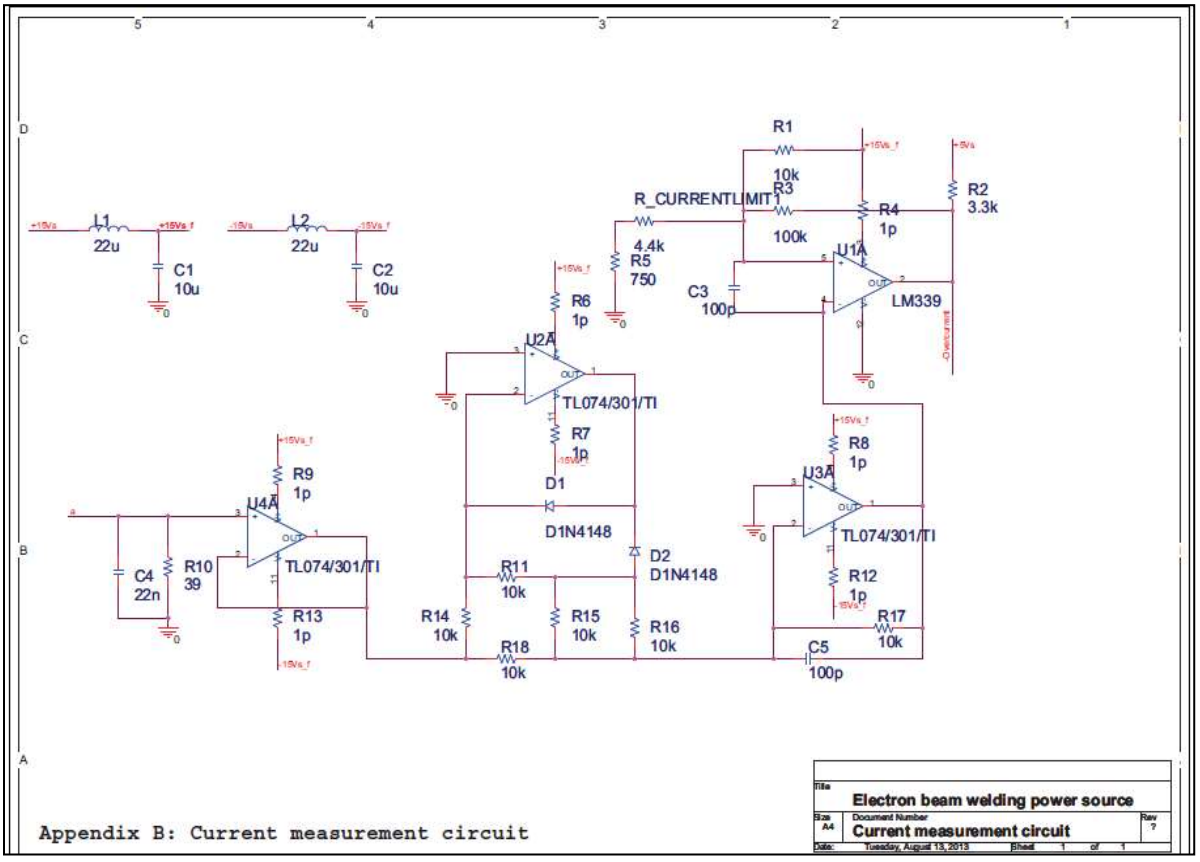


Figure B- 14 Control circuit model: Inverter current measurement circuit

The control circuit also provides overcurrent detection, with overcurrent shutdown. The topology specific circuit model for this circuit is presented in Figure B- 14. The inverter current is sensed by the current shunt depicted in Figure B- 4 at a scale 5000:1 and is fed back into this

circuit. The input stage of this circuit is configured in a way to produce 3.9V at the output of the operational amplifier circuit formed by U4A when the inverter current is 500A.

Current setting in A	Threshold voltage in V	Resistor value of $R_{CURRENTLIMIT\ in\ \Omega}$
150	1.17	9.6E+01
200	1.56	4.1E+02
300	2.34	1.1E+03
400	3.12	1.9E+03
450	3.51	2.3E+03
500	3.9	2.8E+03
550	4.29	3.3E+03
600	4.68	3.8E+03
650	5.07	4.4E+03

Table B- 4 Current limit threshold values

$$(R_x + 750) = \frac{V_x \times 10 \times 10^3}{(15 - V_x)} \qquad \textbf{Equation B- 1}$$

The overcurrent trip is set by the comparator circuit formed by U1A and its associated resistors R1, $R_{CURRENTLIMIT}$ and R5. Adjusting the value of resistor $R_{CURRENTLIMIT}$ allows changing the current trip threshold level. Typical overcurrent level settings and their corresponding threshold voltage values are given in Table B- 4. This table also shows the corresponding resistor values that can be calculated using Equation B- 1.

e. Fault detection circuit

The circuit models for this circuit are presented in Figure B- 15 and Figure B- 16. These circuits act upon the detection of an overcurrent situation. When an overcurrent signal is triggered by the inverter current measurement circuit for ten consecutive switching cycles (duration of 1 ms) this circuit assert a flashover signal. This time is set by the RC network formed by the capacitor C10 and the equivalent resistance of value of the parallel combination of

R42 and R42 (0.69 x 100 nF x 14.49 Ω). This action subsequently pulls the signal HTDEADSET high (Figure B- 16) and as a consequence, the weld dead time sequence explained in the kV demand circuit is initiated.

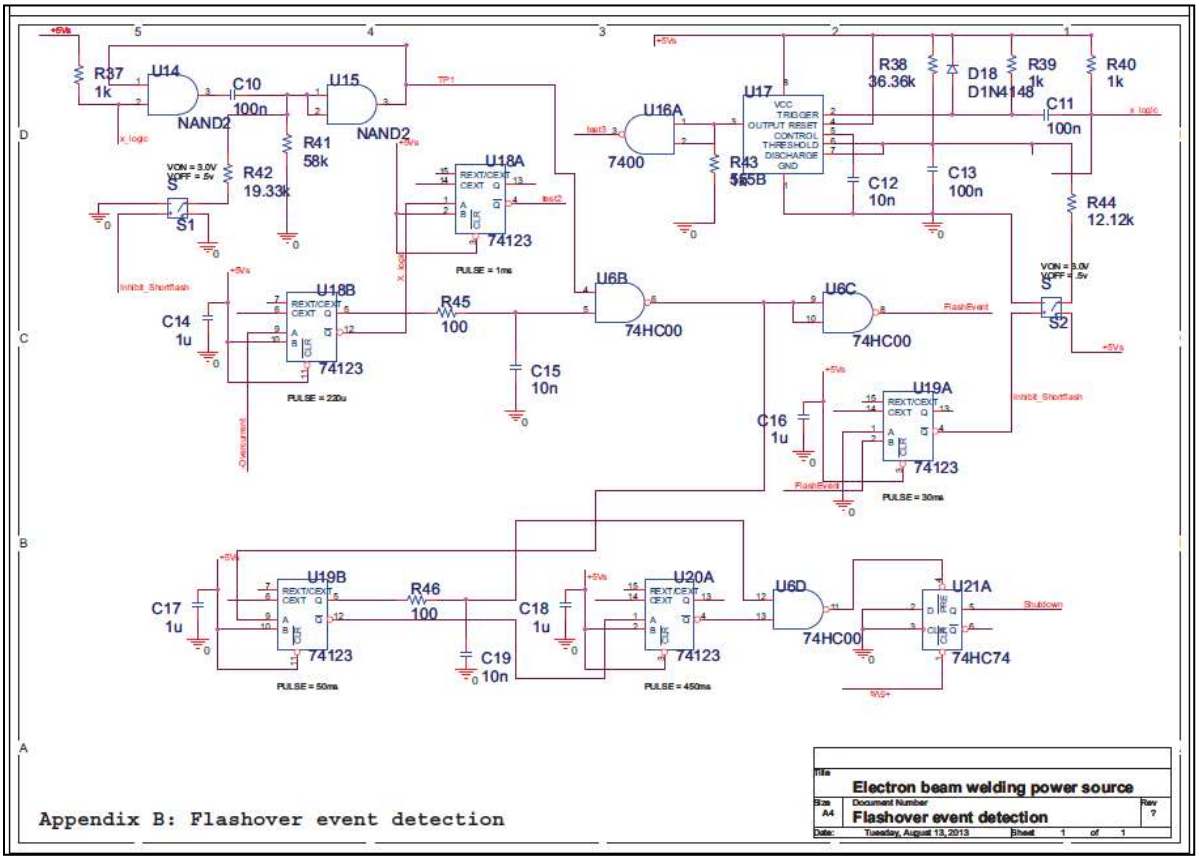


Figure B- 15 Fault detection circuit model: Flashover detection

An inverter current in excess of 650 A is possible at the initial stage of the kV ramping up operation. This overcurrent flow is expected to last for more than 10 switching cycles. This causes the inverter current measurement circuit to assert the overcurrent signal which in a normal operation leads to a flashover detection signal - flashover to be asserted. To avoid false flashover triggering during kV recovery this circuit was implemented with a safeguard signal inhibit_shortflash. This signal prevents retriggering of a flashover signal during the kV recovery

period after flashover, by forcing the signal V_{TP1} to be kept at a logic level '0' for 4 ms (40 switching cycles) on detection of the first overcurrent signal during kV recovery.

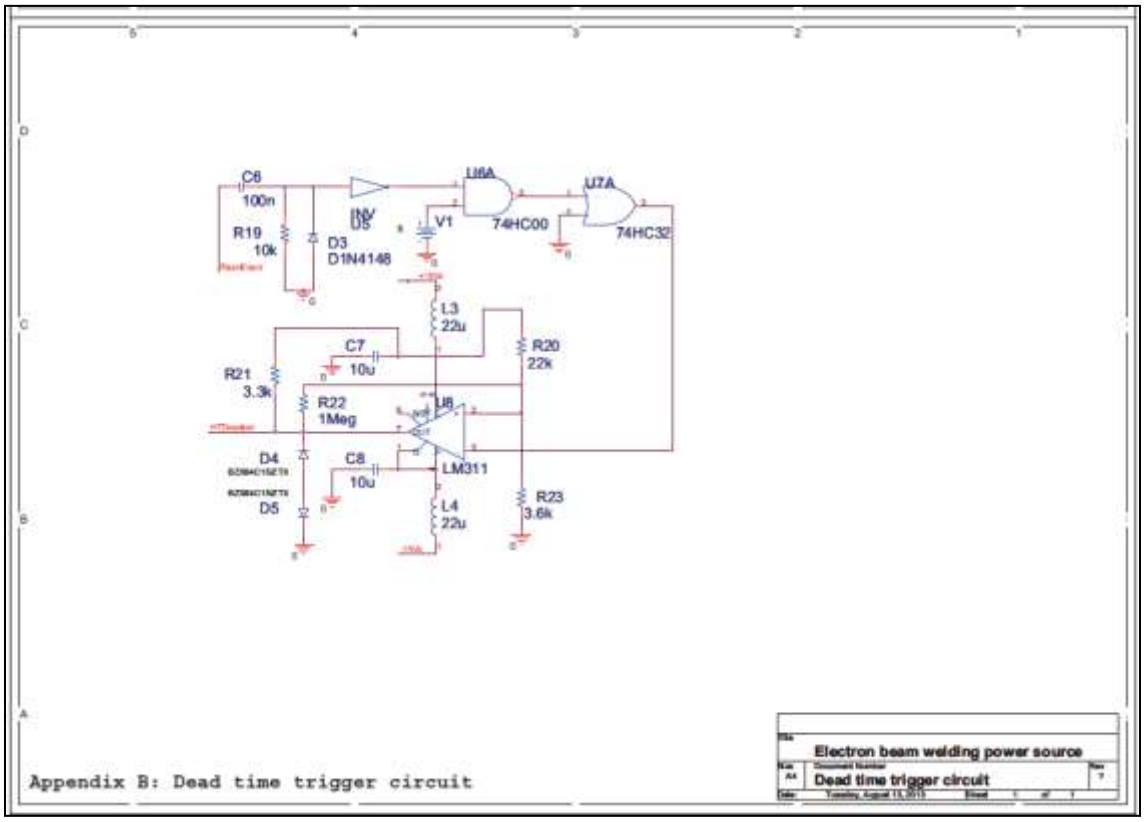


Figure B- 16 Fault detection circuit: Dead time set circuit