



Mechatronics Development and Design for Network Product

Miss Natnari Smitthimedhin

Cooperative Education

Bachelor of Engineering Program in Computer Engineering

Faculty Engineering Major Computer Engineering

Thai – Nichi Institute of Technology

B.E. 2012

Mechatronics Development and Design for Network Product

Miss Natnari Smitthimedhin

Cooperative Education

Bachelor of Engineering Program in Computer Engineering

Faculty Engineering Major Computer Engineering

Thai – Nichi Institute of Technology

B.E. 2012

Evaluation committee

..... **Head of Committee**

(Mr. Prawet Ueatrongchit)

..... **Committee**

(Miss Sidshchadhaa Aumted)

..... **Advisor**

(Dr. Warakorn Srichavengsup)

..... **Head of Cooperative Program**

(Dr. Warakorn Srichavengsup)

Project Name	Mechatronics Development and Design for Network Product
Author	Miss Natnari Smitthimedhin
Faculty	Engineering
Major	Computer Engineering
Advisor Teacher	Dr. Warakorn Srichavengsup
Advisor staff	Miss Tomoyo Murai
Organization Name	NEC Corporation
Business Type	Provider of Electronic and information technology (IT) services and products

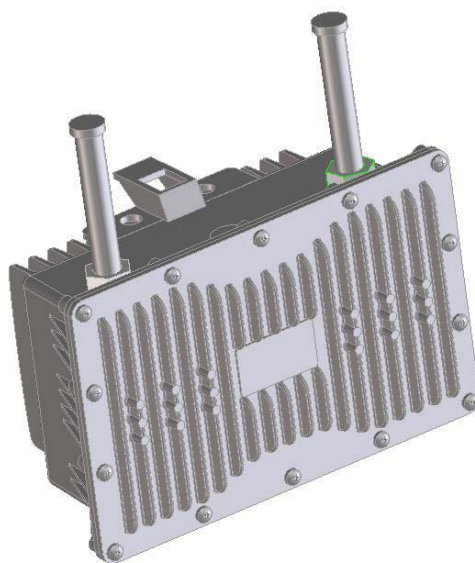
Abstract

The process of mechatronics development and design for network product begin with researching and comparing properties of various network products. Internship student have to draw and analyze 3D models from case studies and do an environment test.

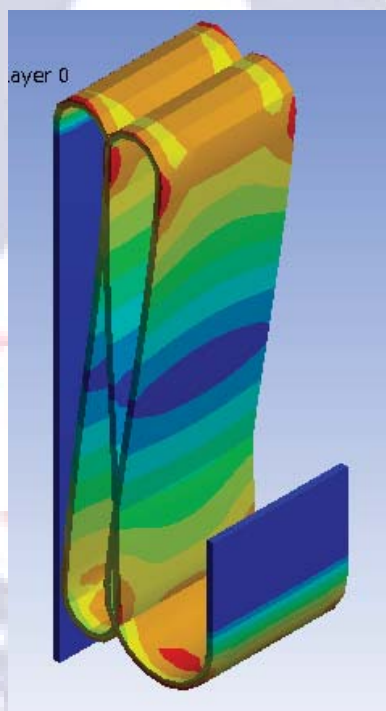
After learning and practicing, work can be summarized as follows

1. Using Creo Elements/Direct 17.0 Modeling to design and draw 3D models.
2. Using ANSYS Workbench platform to analyze 3D models.
3. Making installation manual for network products.
4. Do an environment testing such as IP test and Vibration test, to test sampling product in various case of environment.
5. Writing report of test results.

As internship process, expected result of work is that internship student can further knowledge and skills of drawing and analysis 3D models by software and understand about environment test.



Example of assigned work of drawing 3D models



Example of assigned work of 3D models analysis

Acknowledgment

As 42 days of internship period from August 27, 2012 to October 5, 2012. I have learned and experienced so many things at NEC Corporation. I have always received very good support from many people, especially from my advisors.

1. I would like to express my sincere thanks to my advisor staff, Miss Tomoyo Murai for her very good support. I am most grateful for her teaching and kindly advice, not only the work methodologies but also many other methodologies in life.

2. I am grateful for my advisor teachers of Thai-Nichi Institute of Technology, Dr.Warakorn Srichavengsup for his advice and suggestions.

The special thanks go to NEC Corporation who gave a really good chance for me. I would like to thanks members of network carrier department for their very good support and always taking care of me during my internship period. In addition, I most gratefully acknowledge my parents, my friends and everyone who has facilitated successful completion of a project. To those of you who I did not specifically name, I also give my thanks for moving me towards my goal.

Miss Natnari Smitthimedhin

Author

October 2, 2012

Contents

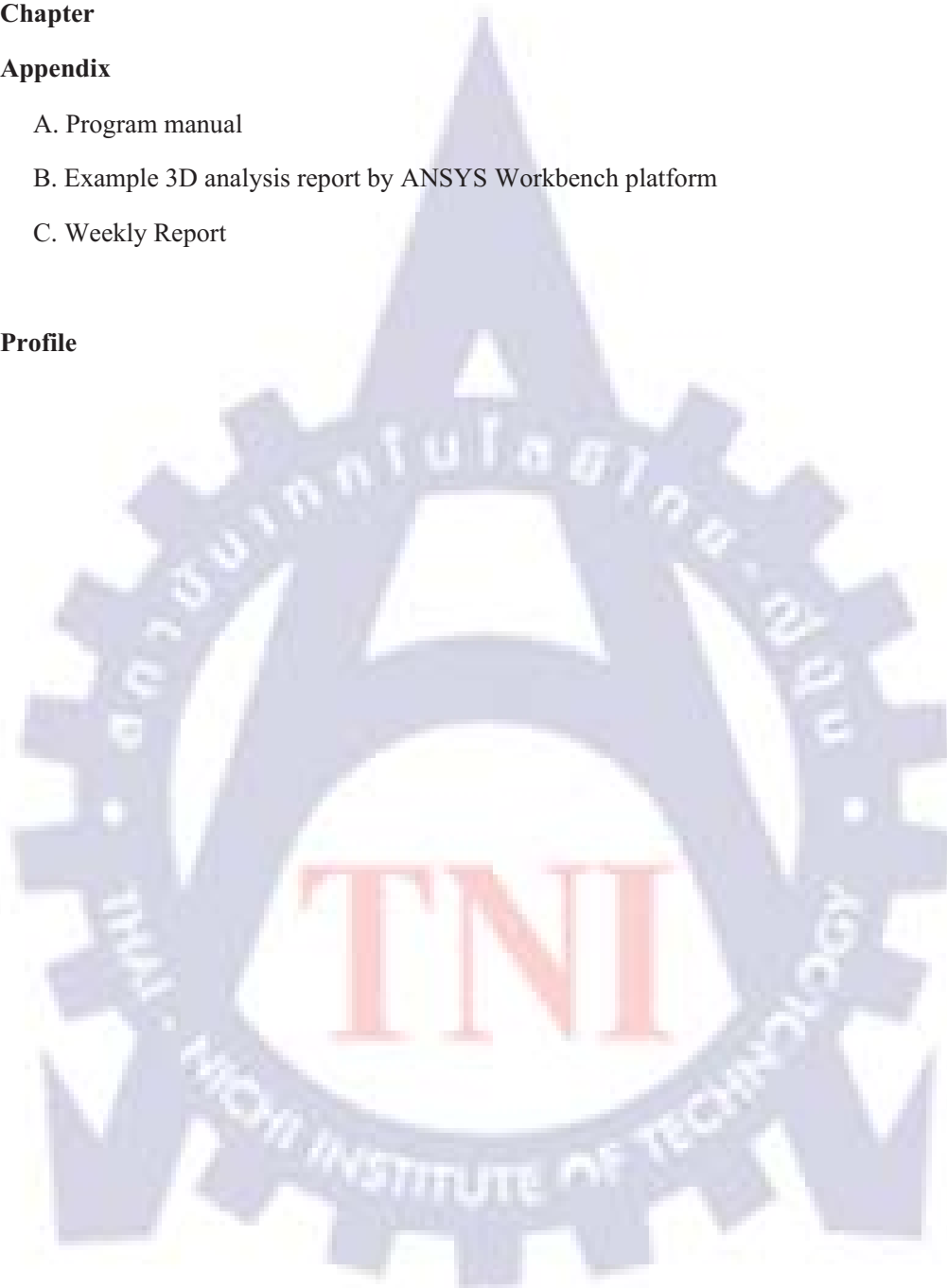
	Page
Abstract	a
Acknowledgment	c
Contents	d
List of Tables	g
List of Figures	i
 Chapter	
1. Introduction	1
1.1 Organization Name and Address	1
1.2 Business Type and Main Business	2
1.3 Organization Management	5
1.4 Position and Assigned Work	6
1.5 Advisor Staff	7
1.6 Work period	7
1.7 Work Purpose	7
1.8 Expected Result	7
 2. Operation Theory and Technology	8
2.1 3D Modeling Technology	8
2.2 Model Analysis Technology	10
2.3 Standard	15
2.3.1 IP code (Ingress Protection Rating or International Protection Rating)	15
2.3.2 JIS (Japanese Industrial Standards)	24
 3. Work plan and procedure	26

Contents (Continue)

Chapter	Page
3. Work plan and procedure	26
3.1 Work plan	26
3.2 Work details	26
3.3 Work procedure	27
3.3.1 Orientation	27
3.3.2 Product research	27
3.3.3 Setup 3D CAD Training, SIM Training	27
3.3.4 Environment Test	29
3.3.5 House Keeping	30
4. Working Procedure and Analyze	31
4.1 Workflow Conclusions	31
4.2 Analysis Results	32
4.2.1 Result of learning Creo Elements/Direct 17.0 Modeling.	32
4.2.2 Result of learning ANSYS Workbench platform	39
4.2.3 Result of writing and making network product installation manual	59
4.2.4 Result of learning about environment testing	59
4.3 Comparing result of work with working purpose	62
5. Conclusions and Suggestions	63
5.1 Conclusions	63
5.2 Work Conclusions	63
5.3 Problems	64
5.4 Suggestions	64
References	65

Contents (Continue)

Chapter	Page
Appendix	
A. Program manual	67
B. Example 3D analysis report by ANSYS Workbench platform	79
C. Weekly Report	97
Profile	104



List of Tables

Table	Page
2.1 Level of protection against foreign objects for each first characteristic numeral level.	16
2.2 Level of protection against foreign objects for each first characteristic numeral level (continue)	17
2.3 Level of protection against foreign objects for each first characteristic numeral level (continue)	18
2.4 Level of protection of the equipment inside the enclosure against harmful ingress of water for each first characteristic numeral level.	18
2.5 Level of protection of the equipment inside the enclosure against harmful ingress of water for each first characteristic numeral level. (continue)	19
2.6 Level of protection of the equipment inside the enclosure against harmful ingress of water for each first characteristic numeral level. (continue)	20
2.7 Level of protection of the equipment inside the enclosure against harmful ingress of water for each first characteristic numeral level. (continue)	21
2.8 Level of protection against access to hazardous parts by persons.	21
2.9 Level of protection of the device.	22
2.10 Example of Vibration test standards	25
3.1 Work plan	26
B.1 Project	79
B.2 Units	79
B.3 Model (A4) > Geometry	80
B.4 Model (A4) > Geometry (Continue)	81
B.5 Model (A4) > Geometry > Parts	82
B.6 Model (A4) > Geometry > Parts (Continue)	83
B.7 Model (A4) > Coordinate Systems > Coordinate System	83
B.8 Model (A4) > Mesh	84

List of Tables

Table	Page
B.9 Model (A4) > Mesh (Continue)	85
B.10 Model (A4) > Analysis	85
B.11 Model (A4) > Analysis (Continue)	86
B.12 Model (A4) > Static Structural (A5) > Analysis Settings	86
B.13 Model (A4) > Static Structural (A5) > Analysis Settings (Continue)	87
B.14 Model (A4) > Static Structural (A5) > Analysis Settings (Continue)	87
B.15 Model (A4) > Static Structural (A5) > Analysis Settings (Continue)	88
B.16 Model (A4) > Static Structural (A5) > Loads	88
B.17 Model (A4) > Static Structural (A5) > Solution	89
B.18 Model (A4) > Static Structural (A5) > Solution (A6) > Solution Information	90
B.19 Model (A4) > Static Structural (A5) > Solution (A6) > Results	90
B.20 Model (A4) > Static Structural (A5) > Solution (A6) > Results (Continue)	91
B.21 Structural Steel > Constants	93
B.22 Structural Steel > Compressive Ultimate Strength	94
B.23 Structural Steel > Compressive Yield Strength	94
B.24 Structural Steel > Tensile Yield Strength	94
B.25 Structural Steel > Tensile Ultimate Strength	94
B.26 Structural Steel > Isotropic Secant Coefficient of Thermal Expansion	94
B.27 Structural Steel > Isotropic Relative Permeability	94
B.28 Structural Steel > Alternating Stress Mean Stress	95
B.29 Structural Steel > Strain-Life Parameters	95
B.30 Structural Steel > Isotropic Elasticity	95

List of Figures

Figure	Page
1.1 Logo of NEC Corporation	1
1.2 NEC Head office building	1
1.3 NEC Tamagawa plant building	2
1.4 NEC Tamagawa plant map	2
1.5 NEC Organization Structure	5
2.1 Logo of PTC Creo elements/direct modeling	8
2.2 PTC Creo elements/direct modeling start up	9
2.3 Example model by Creo elements/direct modeling	10
2.4 Logo of ANSYS Workbench platform	11
2.5 Example model analysis by ANSYS Workbench platform	15
3.1 Comparing properties of network product	27
3.2 3D bracket by Creo Elements/Direct	28
3.3 3D analysis by ANSYS Workbench platform	28
3.4 Design and Simulate 3D analysis	29
3.5 IP test process	29
3.6 Vibration test process	30
4.1 Create workplane	31
4.2 Draw guides	32
4.3 Draw profile	32
4.4 Machine into 3D geometry	32
4.5 Modify geometry	33
4.6 Result of Modify geometry	33
4.7 Add new workplane	33
4.8 Draw guidelines and profile on new workplane	34
4.9 Modify object	34
4.10 Result of Modify object	34

List of Figures (Continue)

Figure	Page
4.11 Select Object to generate a drawing	35
4.12 Select viewto generate a drawing	35
4.13 Generate a drawing	35
4.14 Define details of drawing	36
4.15 Save file	36
4.16 Assembled objects	37
4.17 Position assembled objects	37
4.18 Result ofpositioning assembled objects	37
4.19 Generate drawing ofassembled objects	38
4.20 Define details of drawing ofassembled objects	38
4.21 Coloringobject	38
4.22 Select material for object	39
4.23 Example of assigned work using Creo Elements/Direct	39
4.24 ANSYS Workbench platform start up	40
4.25 Activate ANSYS Workbench	40
4.26 Import the geometry	41
4.27 Select file to import the geometry	41
4.28 Activate Mechanical	42
4.29 Select material to confirm	42
4.30 Confirm the material	42
4.31 Click the Engineering data tab	43
4.32 Select Engineering Data Sources	43
4.33 Select wanted data source	43
4.34 Select wanted source	44
4.35 Confirm Engineering Data Sources	44
4.36 Return to Project	44

List of Figures (Continue)

Figure	Page
4.37 Set up Unit	44
4.38 Define force on object	45
4.39 Click “Force”	45
4.40 Select the inner face to define force	45
4.41 Confirm force	46
4.42 Change Define by	46
4.43 Change value of Coordinate System	46
4.44 Confirm the force direction	47
4.45 Define Fix part on object	47
4.46 Select 2 faces to fix	47
4.47 Confirm Fix part on object	48
4.48 Fix part on object is confirmed	48
4.49 Select Solution	48
4.50 Select Equivalent(von-Mises)	48
4.51 Select Vector Principal	49
4.52 Select Total	49
4.53 Solve button	49
4.54 Analysis progress	49
4.55 Select Equivalent Stress	50
4.56 Contour is displayed	50
4.57 Result of Minimum and Maximum	50
4.58 New section plane button	51
4.59 Define the direction of the section	51
4.60 The section is created	51
4.61 Rotate button	51
4.62 Rotate the geometry	52

List of Figures (Continue)

Figure	Page
4.63 Select the slice Plane	52
4.64 Move section plane	52
4.65 Select the slice Planes	52
4.66 Result of modifying slice plane	53
4.67 Select Vector Principal Stress	53
4.68 Vector of principal stress is displayed	54
4.69 Select Total deformation	54
4.70 Contour of total deformation is displayed	54
4.71 Select Show Undeformed Model	55
4.72 Undeformed Model is displayed	55
4.73 Select contour	55
4.74 Select Equivalent Stress	55
4.75 Click the play button	56
4.76 Select type of solution	56
4.77 Figure is inserted	56
4.78 Figure is inserted anywhere user wanted	56
4.79 Select Report Preview	57
4.80 Report in progress	57
4.81 Save the report	57
4.82 Send report to other program	58
4.83 Example of assigned work using ANSYS Workbench platform	58
4.84 Example of assigned work of writing and making installation manual	59
4.85 Example of assigned work of writing and making installation manual using Creo Elements/Direct	59
4.86 Example of assigned work of IP test	60
4.87 Example of report of IP test	60

List of Figures (Continue)

Figure	Page
4.88 Example of assigned work of Vibration test	61
4.89 Example of checking result of Vibration test	62
4.90 Example of report of Vibration test	62
A.1 Workplane tool	67
A.2 Create more Workplane tool	67
A.3 Modify Workplane tool	68
A.4 Create 2D tool	68
A.5 Contour tool	69
A.6 Modify 2D tool	69
A.7 2D Geometry tool	70
A.8 Machine tool	71
A.9 Modify 3D tool	72
A.10 Assembly tool	73
A.11 Modify part and assembly tool	74
A.12 Select Application	74
A.13 Select Annotation	74
A.14 Create Drawing tool	74
A.15 Create Drawing parameters	75
A.16 Select Add Views to add views	75
A.17 Select Application	76
A.18 Select Rendering	76
A.19 Rendering Browser Windows	76
A.20 Result of Rendering	77
B.1 Project	79
B.2 Model (A4) > Static Structural (A5) > Force	89
B.3 Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Stress > Figure	92

List of Figures (Continue)

Figure	Page
B.4 Model (A4) > Static Structural (A5) > Solution (A6) > Vector Principal Stress > Figure	92
B.5 Model (A4) > Static Structural (A5) > Solution (A6) > Total Deformation > Figure	93



Chapter 1

Introduction

1.1 Organization Name and Address

Company Name

NEC Corporation



Figure 1.1 Logo of NEC Corporation

Location

Head Office: 7-1, Shiba 5-chome, Minato-ku, Tokyo 108-8001 Japan



Figure 1.2 NEC Head office building

Tamagawa Plant: 1753 Shimonumabe, Nakahara-ku, Kawasaki, Kanagawa 211-8666, Japan



Figure 1.3 NEC Tamagawa plant building

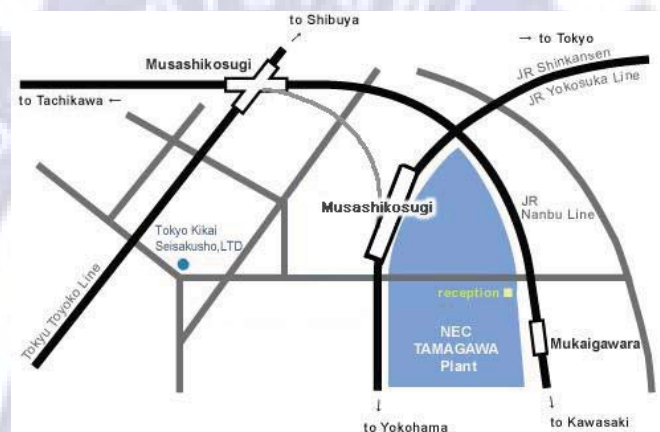


Figure 1.4 NEC Tamagawa plant map

1.2 Business Type and Main Business

1.2.1 Business Outline

1.2.1.1 IT Solutions Business

NEC supplies government agencies and private-sector companies with a range of IT services covering system building, maintenance and support, operation and outsourcing, as well as cloud services. NEC also supplies the platforms essential to the building of IT systems and enterprise network systems, such as servers, storage, software and IP telephony systems.

1.2.1.2 Carrier Network Business

NEC supplies equipment required in network implementation to carriers, along with network control platform systems and operating services. NEC's wealth of experience in large-scale network implementation and strong technical capabilities contribute to the development of highly reliable communications networks.

1.2.1.3 Social Infrastructure Business

NEC supplies industrial systems that support social infrastructure, including broadcasting systems, artificial satellites and integrated CCTV surveillance systems for airports and local governments, as well as other security-related systems that enhance public safety and security.

1.2.1.4 Personal Solutions Business

NEC supplies individuals and private-sector companies with mobile handsets, personal computers, Internet services as well as monitors. It is also engaged in the creation of new devices and services for the cloud computing era.

1.2.2 Products

NEC provides electronics, information technology (IT) and network solutions to business enterprises, communications services providers and to government agencies as follows

1.2.2.1 Display Solutions

- Monitors and Projectors (NEC Display Solutions, Ltd.) Digital Cinema

1.2.2.2 Computing Products

- Servers
- Disk storage
- Tape Storage
- Super Computers (Thin Client, POS Systems, Industrial Computers)
- Personal Computers (Desktop PCs, Notebooks, Workstations)

1.2.2.3 Software

- Middleware
- Embedded System Design
- Business Applications

1.2.2.4 Telecom Network

1.2.2.5 Enterprise Network and Communication

- Communication Platforms
- Software Based Communications

1.2.2.6 Social Infrastructure

- Digital Broadcast
- Postal Automation

1.2.2.7 Electron Devices

- Energy Devices, Network Devices and Functional Devices (NEC TOKIN Corporation.)
- Fiber Optic Devices
- Display Devices (NEC LCD Technologies, Ltd.)
- Uncooled infrared detectors

1.3 Organization Management

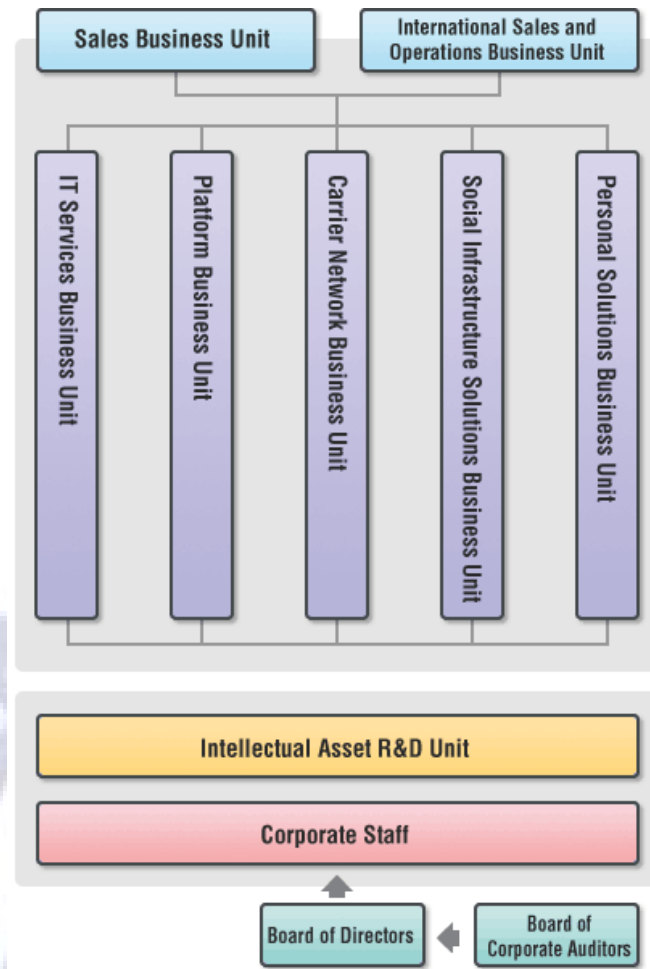


Figure 1.5 NEC Organization Structure

Summary of each unit

1.3.1 Sales Business Unit

- Sales operations covering all domestic customers and sales operations encompassing NEC's entire product line-up
- Sales operations carried out in collaboration with domestic partner companies

1.3.2 International Sales and Operations Business Unit

Sales operations covering all overseas customers and encompassing NEC's entire line-up of products and services based on cooperation with overseas subsidiaries

1.3.3 IT Services Business Unit

IT service business (including consulting, SI construction, operation, maintenance, outsourcing and more)

1.3.4 Platform Business Unit

Software product business (including OS, SI platform software and more), software support service business, enterprise network solution business and product business (including IT/Network integrated platform, open servers, mainframes, super computers, storage, industry specific work stations and more)

1.3.5 Carrier Network Business Unit

Network integration and product businesses aimed at telecommunications carriers

1.3.6 Social Infrastructure Solutions Business Unit

Social infrastructure solutions business (including broadcasting, control, aerospace, defense and other systems)

1.3.7 Personal Solutions Business Unit

- Product business (including mobile terminals, mobile software and more)
- Product business (including PCs, displays, peripheral devices and more), service business and -BIGLOBE service business

1.3.8 Intellectual Asset R&D Unit

Research and development aimed at strengthening the NEC Group's technical competence, the development of new businesses and business models and the creation of intellectual assets and the enhancement of their profits

1.4 Position and Assigned Work

Position: Internship Student of Network Platform Development Division of Carrier Network Business Unit(2nd Basic Technologies Development Group).

Assigned Work: 1. 3D Modeling

2. Mechanical Analysis

3. Environment Test

1.5 Advisor Staff

Advisor staff name: Miss Tomoyo Murai

Position: Assistant Manager

Tel.: +81 44 455 8391

E-mail: t-murai@ap.jp.nec.com

1.6 Work period

August 27, 2012 – October 5, 2012

1.7 Work Purpose

- 1.7.1 To learn to develop mechatronics for network product by case study.
- 1.7.2 To learn to design mechatronics for network product by case study.
- 1.7.3 To further knowledge of drawing 3D CAD model by using Creo Elements/Direct 17.0 Modeling.
- 1.7.4 To learn to analyze 3D model by using ANSYS Workbench platform.
- 1.7.5 To learn about environment test.

1.8 Expected Result

- 1.8.1 Be able to develop mechatronics for network product.
- 1.8.2 Be able to design mechatronics for network product.
- 1.8.3 Understand the explicit modeling approach of Creo Elements/Direct 17.0 Modeling.
- 1.8.4 Can draw 3D CAD model by using Creo Elements/Direct 17.0 Modeling.
- 1.8.5 Can analyze 3D model by using ANSYS Workbench platform.
- 1.8.6 Understand about environment test.

Chapter 2

Operation Theory and Technology

2.1 3D Modeling Technology

Creo elements/direct modeling 17.0

Creo Elements/Direct Modeling is a complete environment for 3D product design. The direct modeling approach of Creo Elements/Direct Modeling makes creating and modifying 3D designs fast, easy and flexible. Many of the world's best known product development companies depend on the Creo Elements/Direct product family for a direct 3D product development process in order to meet shorter design cycles, create one-off product designs quickly, and respond to unpredictable design requirement changes.



Figure 2.1 Logo of PTC Creo elements/direct modeling

The Creo Elements/Direct product family provides a lightweight 3D design approach that can help in many engineering processes, including:

- Concept development
- Verification and validation
- System design
- Design outsourcing
- Detailed design

Creo Elements/Direct delivers speed, flexibility, and responsiveness-to-change for customers pursuing a lightweight design strategy. When designing in 3D with Creo Elements/Direct, models stay lightweight and flexible through direct, “on-the-fly” geometry interactions. Product designs evolve freely without designers needing to anticipate downstream design changes or maintain embedded design logic such as tangency conditions. Creo

Elements/Direct Modeling provides flexibility in several other ways, by giving the ability to address unexpected and radical changes to product designs, respond to changes in design team composition, and work easily with design data from any 3D or 2D CAD system as if it were native design data.

Creo Elements/Direct Modeling, with its direct approach to 3D design, is faster and easier to learn and use than other 3D CAD systems. In addition, Creo Elements/Direct offers the simplicity of 2D with the power of 3D.

Consequently, direct modeling files are, on average, one-third the size of parametric modeling files, which makes models easier to access, modify, archive, and share. As a member of the Creo product family, Creo Elements/Direct users can apply downstream design, simulation and visualization capabilities provided in Creo apps, such as Creo Parametric, Creo View MCAD, and Creo Simulate. The extended capabilities provided in Creo further enable fast, flexible, and powerful designs. As part of PTC's integral Product Development System (PDS), Creo Elements/Direct gives a seamless user experience with other PTC product families, including Creo Elements/View, Arbortext®, Mathcad®, Creo Elements/Direct, and Windchill.

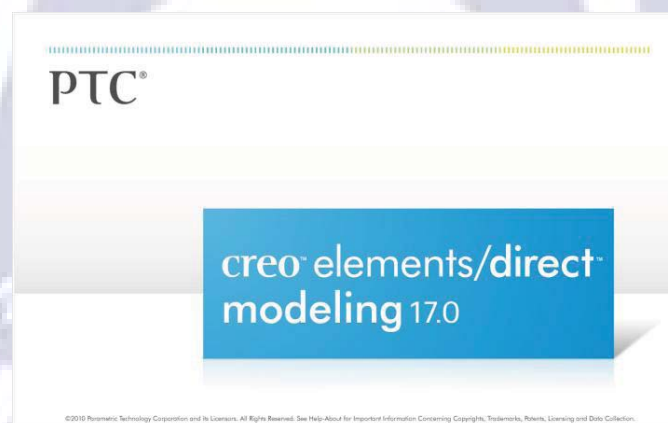


Figure 2.2 PTC Creo elements/direct modeling start up

From all above, this allow quickly incorporate downstream 3D design changes and radically repurpose designs including cutting, copying, and pasting geometry which makes the initial training and overall learning curve on Creo Elements/Direct products fast and easy.

CREO elements/direct Features & Benefits

- Powerful and flexible parametric 3D CAD capabilities enable superior product differentiation and manufacturability
- Broad range of concept design capabilities facilitates rapid new product introduction
- Seamless data interoperability and a common user experience across apps and extensions allow you to develop everything from concept to manufacturing faster and cheaper
- Ability to accommodate late stage changes and automatic propagation of design changes to all downstream deliverables allows you to design with confidence
- Automated generation of associative manufacturing and service deliverables accelerates time-to-market and reduces cost

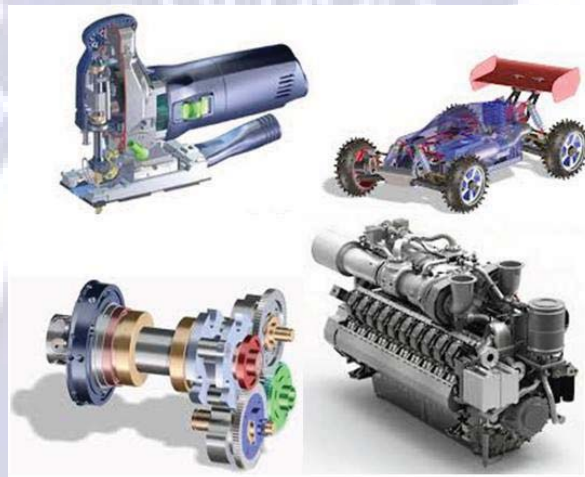


Figure 2.3 Example model by Creo elements/direct modeling

2.2 Model Analysis Technology**ANSYS Workbench platform**

ANSYS, Inc. is an engineering simulation software (computer-aided engineering, or CAE) developer that is headquartered south of Pittsburgh in Canonsburg, Pennsylvania, United States. ANSYS was listed on the NASDAQ stock exchange in 1996. In late 2011, Investor's Business Daily ranked ANSYS as one of only six technology businesses worldwide to receive the highest possible score on its Smart Select Composite Ratings. ANSYS has been recognized as a

strong performer by a number of other sources as well. The organization reinvests 15 percent of its revenues each year into research to continually refine the software.

The ANSYS Workbench platform is the framework upon which the industry's broadest and deepest suite of advanced engineering simulation technology is built. An innovative project schematic view ties together the entire simulation process, guiding the user through even complex multi physics analyses with drag-and-drop simplicity. With bi-directional CAD connectivity, powerful highly-automated meshing, a project-level update mechanism, pervasive parameter management and integrated optimization tools, the ANSYS Workbench platform delivers unprecedented productivity, enabling Simulation Driven Product Development.

The ANSYS Workbench platform automatically forms a connection to share the geometry for both the fluid and structural analyses, minimizing data storage and making it easy to study the effects of geometry changes on both analyses. In addition, a connection is formed to automatically transfer pressure loads from the fluid analysis to the structural analysis.



Figure 2.4 Logo of ANSYS Workbench platform

ANSYS Workbench platform Journaling and Scripting

Operations executed within the ANSYS Workbench platform can be recorded to a Python-based journal that can be replayed to easily recreate an ANSYS Workbench session. Users can modify the journal or write a new set of instructions to create a custom ANSYS Workbench script for automating repetitive tasks or performing complex operations. Journals and scripts can be replayed interactively or in batch mode when ANSYS Workbench is launched. ANSYS Workbench scripting is based on the versatile, object-oriented Python scripting language, which provides a rich set of data types and standard libraries for efficient programming. Furthermore, the Python implementation used in ANSYS Workbench is well integrated with the .NET Framework (WindowsR) and Mono (LinuxR), providing interoperability with popular programs like MicrosoftR ExcelR to access ANSYS Workbench parameters, perform external calculations and drive the overall simulation process. In addition to replaying journal files,

ANSYS Workbench provides a command window to invoke individual commands. Keyboard shortcuts, command- completion and command recall facilitate command window interaction and reduce tedious typing. Many applications hosted in ANSYS Workbench support their own scripting languages (e.g., ANSYSR Mechanical™ APDL), and these application-specific commands can be embedded in an ANSYS Workbench script. This coordination between ANSYS Workbench and its applications provides comprehensive scripting support of the complete engineering simulation process.

ANSYS Workbench Features

- Bidirectional, parametric links with all major CAD systems
- Integrated, analysis-focused geometry modeling, repair, and simplification via ANSYS Design Modeler
- Highly-automated, physics-aware meshing
- Automatic contact detection
- Unequalled depth of capabilities within individual physics disciplines
- Unparalleled breadth of simulation technologies
- Complete analysis systems that guide the user start-to-finish through an analysis
- Comprehensive multi physics simulation with drag-and-drop ease of use
- Flexible components enable tools to be deployed to best suit engineering intent
- Innovative project schematic view allows engineering intent, data relationships, and the state of the project to be comprehended at a glance
- Complex project schematics can be saved for re-use
- Pervasive, project-level parameter management across all physics
- Automated what-if analyses with integrated design point capability
- Adaptive architecture with scripting and journaling capabilities and API's enabling rapid integration of new and third-party solutions

The applications hosted in the ANSYS Workbench environment support parametric variations, including CAD geometry dimensions, material properties, boundary conditions and derived results. Parameters defined within the applications can be managed directly from the project window, making it easy to investigate multiple variations of an analysis. From within the

project window, a series of design points can be defined and executed to complete a what-if study with a single operation.

To fully leverage the power of parametric analysis, ANSYS DesignXplorer software extends the ANSYS Workbench platform to drive Design of Experiments, goal-driven optimization, min/max search or even perform Six Sigma analysis to investigate design robustness. All of this power is available across all applications, all physics and all solvers available within the ANSYS Workbench environment, including ANSYS Mechanical APDL.

ANSYS DesignXplorer

ANSYS DesignXplorer was developed to leverage the power of ANSYS Workbench for parametric analyses. ANSYS Workbench makes it easy to create and manage parameters across a wide range of ANSYS products, has a persistent setup and makes automatic updates. ANSYS DesignXplorer takes advantage of these ANSYS Workbench strengths and enables to explore, understand and optimize design so simulation can be applied to drive product development.

ANSYS DesignModeler

All engineering simulation starts with geometry to represent the design. This could be a solid component for a structural analysis or the air volume for a fluid or electromagnetic study. This geometry has been produced in a computer-aided design (CAD) system or built from scratch. ANSYS DesignModeler software is the gateway to geometry handling for analysis with software from ANSYS.

ANSYS DesignModeler has connections to all major CAD systems, allowing seamless transfer of existing data including parameters. The parameters can then be adjusted and the design updated, and any feature removal or simplification is maintained. This results in rapid turnaround of any design changes and updates.

ANSYS DesignModeler technology also provides powerful tools for construction of geometry from the ground up. A complex model can be produced using familiar solid modeling operations. Built on the Parasolid kernel, the geometry engine is robust and conforms to industry standards. Two-dimensional sketches can be extruded into 3-D solids and then modified with Boolean operations. A construction history is recorded during the creation of the geometry,

allowing the user to make changes and then update the design. Whether importing an existing CAD model or building one, the ANSYS DesignModeler product is fully parametric. Combine this with parametric meshing and parametric solver setup within the ANSYS Workbench platform and the same geometry can be used for multiple design variations.

ANSYS Meshing

Mesh generation is one of the most critical aspects of engineering simulation. Too many cells may result in long solver runs, and too few may lead to inaccurate results. ANSYS Meshing technology provides a means to balance these requirements and obtain the right mesh for each simulation in the most automated way possible. ANSYS Meshing technology has been built on the strengths of stand-alone, class-leading meshing tools. The strongest aspects of these separate tools have been brought together in a single environment to produce some of the most powerful meshing available. The highly automated meshing environment makes it simple to generate the following mesh types:

- Tetrahedral
- Hexahedral
- Prismatic inflation layer
- Hexahedral inflation layer
- Hexahedral core
- Body fitted Cartesian
- Cut cell Cartesian

Consistent user controls make switching methods very straight forward and multiple methods can be used within the same model. Mesh connectivity is maintained automatically. Different physics requires different meshing approaches. Fluid dynamics simulations require very high-quality meshes in both element shape and smoothness of sizes changes. Structural mechanics simulations need to use the mesh efficiently as run times can be impaired with high element counts. ANSYS Meshing has a physics preference setting ensuring the right mesh for each simulation.

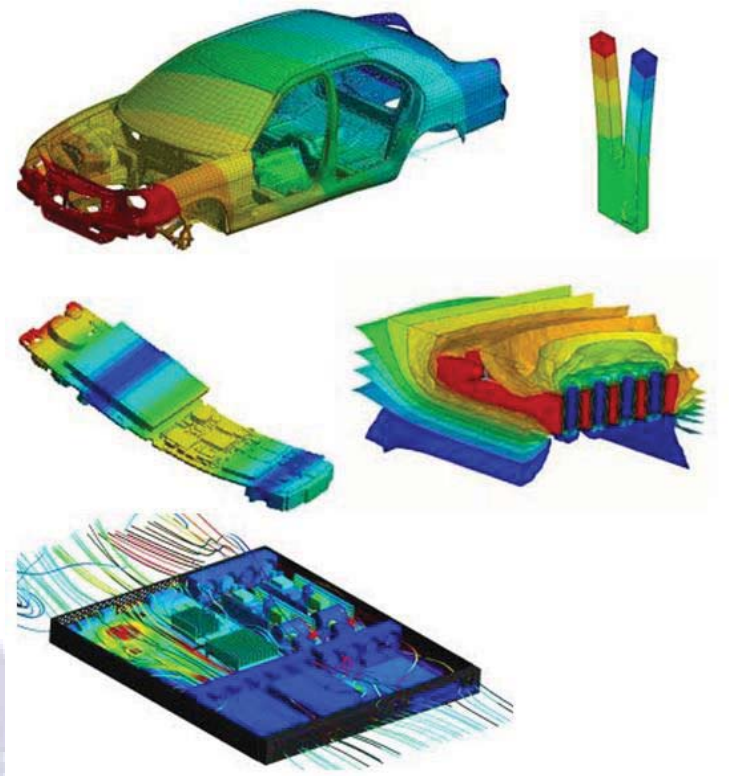


Figure 2.5 Example model analysis by ANSYS Workbench platform

2.3 Standard

2.3.1 IP code (Ingress Protection Rating or International Protection Rating)

As defined in international standard IEC 60529, It is a coding system to indicate the degrees of protection provided by an enclosure against access to hazardous parts, ingress of solid foreign objects (including body parts like hands and fingers), dust, accidental contact, ingress of water and to give additional information in connection with such protection.

IP Code is a code of the letters IP followed by two digits or one digit and one letter and an optional letter. Measures to protect both the enclosure and the equipment inside the enclosure against external influences or conditions such as

- mechanical impacts
- corrosion
- corrosive solvents (for example, cutting liquids)

- fungus
- vermin
- solar radiation
- icing
- moisture (for example, produced by condensation)
- explosive atmospheres

and the protection against contact with hazardous moving parts external to the enclosure (such as fans), are matters for the relevant product standard to be protected.

The digits (characteristic numerals) indicate conformity with the conditions summarized in the tables below. Where there is no protection rating with regard to one of the criteria, the digit is replaced with the letter X. For example, an electrical socket rated IP22 is protected against insertion of fingers and will not be damaged or become unsafe during a specified test in which it is exposed to vertically or nearly vertically dripping water. IP22 or 2X are typical minimum requirements for the design of electrical accessories for indoor use.

The first digit of the IP code indicates the degree of protection against solid foreign objects from entering the electrical device

Table 2.1 Level of protection against foreign objects for each first characteristic numeral level.

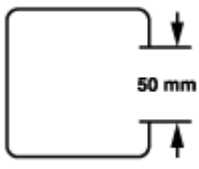
Level	Against ingress of solid foreign	Effective against
0	-	No protection against contact and ingress of objects
1	≥ 50 mm diameter	 Any large surface of the body, such as the back of hand, but no protection against deliberate contact with a body part

Table 2.2 Level of protection against foreign objects for each first characteristic numeral level
(continue)

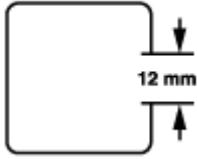
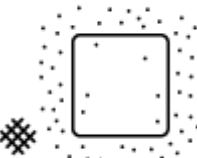
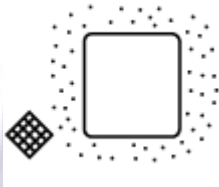
Level	Against ingress of solid foreign	Effective against
2	≥ 12.5 mm diameter	 Fingers or similar objects
3	≥ 2.5 mm diameter	 Tools, thick wires, etc.
4	≥ 1 mm diameter	 Most wires, screws, etc.
5	dust protected	 Ingress of dust is not entirely prevented, but it must not enter in sufficient quantity to interfere with the satisfactory operation of the equipment; complete protection against contact

Table 2.3 Level of protection against foreign objects for each first characteristic numeral level
(continue)

Level	Against ingress of solid foreign	Effective against
6	dust tight	 No ingress of dust; complete protection against contact

The second digit of the IP code indicates the degree of protection against the ingress of various forms of moisture (e.g. drip, spray, submersion, etc.) into the component. Tests to determine the level of protection are carried out with fresh water and do not take into account the use of solvents.

Table 2.4 Level of protection of the equipment inside the enclosure against harmful ingress of water for each first characteristic numeral level.

Level	Protected against	Testing for	Details
0	Not protected		
1	Dripping water	 Dripping water (vertically falling drops) shall have no harmful effect.	Test duration: 10 minutes Water equivalent to 1mm rainfall per minute

Table 2.5 Level of protection of the equipment inside the enclosure against harmful ingress of water for each first characteristic numeral level. (continue)

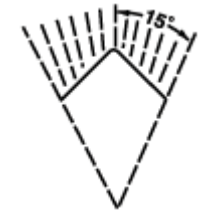
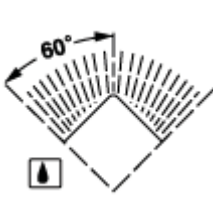
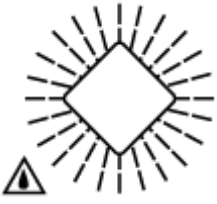
Level	Protected against	Testing for	Details
2	Dripping water when tilted up to 15°	 Vertically dripping water shall have no harmful effect when the enclosure is tilted at an angle up to 15° from its normal position.	Test duration: 10 minutes Water equivalent to 3mm rainfall per minute
3	Spraying water	 Water falling as a spray at any angle up to 60° from the vertical shall have no harmful effect.	Test duration: 5 minutes Water volume: 0.7 litres per minute Pressure: 80–100 kPa
4	Splashing water	 Water splashing against the enclosure from any direction shall have no harmful effect.	Test duration: 5 minutes Water volume: 10 litres per minute Pressure: 80–100 kPa

Table 2.6 Level of protection of the equipment inside the enclosure against harmful ingress of water for each first characteristic numeral level. (continue)

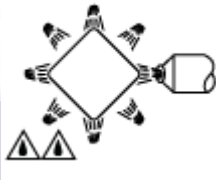
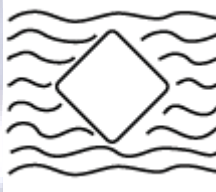
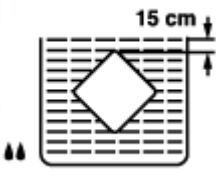
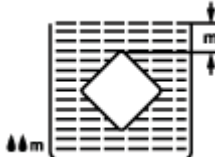
Level	Protected against	Testing for	Details
5	Water jets	 Water projected by a nozzle (6.3mm) against enclosure from any direction shall have no harmful effects.	Test duration: at least 3 minutes Water volume: 12.5 litres per minute Pressure: 30 kPa at distance of 3m
6	Powerful water jets	 Water projected in powerful jets (12.5mm nozzle) against the enclosure from any direction shall have no harmful effects.	Test duration: at least 3 minutes Water volume: 100 litres per minute Pressure: 100 kPa at distance of 3m
7	Immersion up to 1 m	 Ingress of water in harmful quantity shall not be possible when the enclosure is immersed in water under defined conditions of pressure and time (up to 1 m of submersion).	Test duration: 30 minutes Immersion at depth of 1m

Table 2.7 Level of protection of the equipment inside the enclosure against harmful ingress of water for each first characteristic numeral level. (continue)

8	Immersion beyond 1 m	 The equipment is suitable for continuous immersion in water under conditions which shall be specified by the manufacturer. Normally, this will mean that the equipment is hermetically sealed. However, with certain types of equipment, it can mean that water can enter but only in such a manner that it produces no harmful effects.	Test duration: continuous immersion in water Depth specified by manufacturer
---	----------------------	---	--

Additional letters

The standard defines additional letters that can be appended to classify only the level of protection against access to hazardous parts by persons.

Table 2.8 Level of protection against access to hazardous parts by persons.

Level	Protected against access to hazardous parts with
A	Back of hand
B	Finger
C	Tool
D	Wire

Further letters can be appended to provide additional information related to the protection of the device.

Table 2.9 Level of protection of the device.

Letter	Meaning
H	High voltage device
M	Device moving during water test
S	Device standing still during water test
W	Weather conditions

Examples of designations with the IP Code

- IP Code not using optional letters

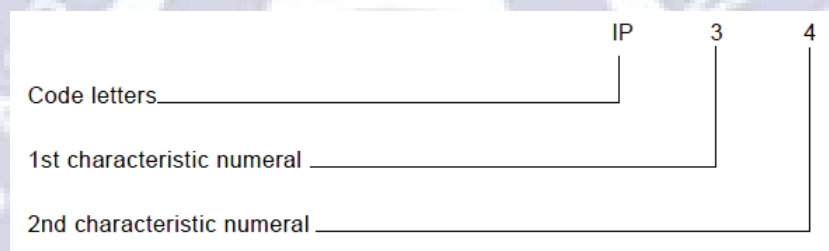


Figure 2.6 IP Code not using optional letters

An enclosure with this designation (IP Code)

- (3) – protects persons, handling tools having a diameter of 2,5 mm and greater, against access to hazardous parts;
- protects the equipment inside the enclosure against ingress of solid foreign objects having a diameter of 2,5 mm and greater;
- (4) – protects the equipment inside the enclosure against harmful effects due to water splashed against the enclosure from any direction.

- IP Code using optional letters

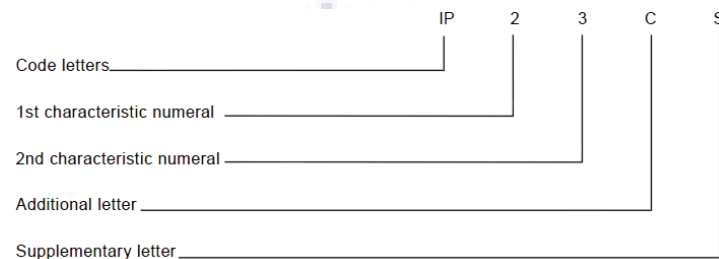


Figure 2.7 IP Code using optional letters

An enclosure with this designation (IP Code)

- (2) – protects persons against access to hazardous parts with fingers;
 - protects the equipment inside the enclosure against ingress of solid foreign objects having a diameter of 12,5 mm and greater;
- (3) – protects the equipment inside the enclosure against the harmful effects due to water sprayed against the enclosure;
- (C) – protects persons handling tools having a diameter of 2,5 mm and greater and a length not exceeding 100 mm against access to hazardous parts (the tool may penetrate the enclosure up to its full length);
- (S) – is tested for protection against harmful effects due to the ingress of water when all the parts of the equipment are stationary.

Examples for the use of letters in the IP Code

The following examples are to explain the use and arrangement of letters in the IP Code.

- IP44 – no letters, no options;
- IPX5 – omitting first characteristic numeral;
- IP2X – omitting second characteristic numeral;
- IP20C – using additional letter;
- IPXXC – omitting both characteristic numerals, using additional letter;
- IPX1C – omitting first characteristic numeral, using additional letter;

IP3XD – omitting second characteristic numeral, using additional letter;

IP23S – using supplementary letter;

IP21CM – using additional letter and supplementary letter;

IPX5/IPX7 – giving two different degrees of protection by an enclosure against both water jets and temporary immersion for “versatile” application.

2.3.2 JIS (Japanese Industrial Standards)

JIS is the standards used for industrial activities in Japan. The standardization process is coordinated by Japanese Industrial Standards Committee (JISC) and published through Japanese Standards Association (JSA). The objective of the Japanese Standards Association - JSA - is "to educate the public regarding the standardization and unification of industrial standards, and thereby to contribute to the improvement of technology and the enhancement of production efficiency". For transportation vibration test, two standards of JIS are used.

JIS Z 0200 (drop and vibration tests)

This standard defines testing procedures for evaluating whether the degree of protection for packaging against falling impact, vibration, and compression which the packaged freights receive in the distribution process is adequate or not. NOTE Dangerous goods, packaged freights of not less than 1000 kg in total mass, and those exceeding 230 cm in maximum dimension of edge or diameter are excluded.

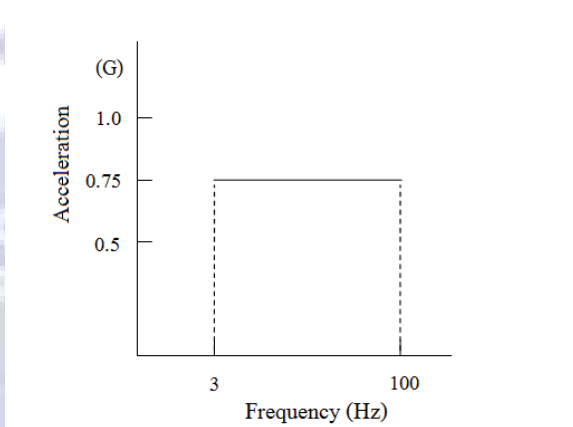
JIS Z 0232/ISO13355: Packaged freights-Method of vibration test

This standard is the Japanese Industrial Standard prepared by translating ISO 8318: 2000, Packaging-Complete, filled transport packages and unit loads - Sinusoidal vibration tests using a variable frequency, and ISO 13355: 2001, Packaging-Complete, filled transport packages and unit loads-Vertical random vibration test, and by making some minor changes in technical contents. This standard specifies both a vertical random vibration test and a sinusoidal sweep vibration test. Since the vertical random vibration is the most appropriate method to create the similar vibration conditions during transportation, the priority should be placed on the vertical random vibration test, not on other kinds of tests including a sinusoidal logarithmic sweep

vibration test, given the condition that the testing equipment is not available. For your reference, the two former standards: sinusoidal uniform sweep vibration tests and sinusoidal constant vibration tests, are enclosed in Appendix.

Test shall be conducted according to the test standard.

Table 2.10 Example of Vibration test standards

Method	Test standards	Application
Test with vibration tester	[JIS Z 0200] [JIS Z 0232]  Vibration time - Transportation distance 1. Below 1000km.: 20 min. 2. 1000 km. to 2000 km.: 40 min. 3. 2000 km. or over: 60 min. (Distance by ship or aircraft not included.) Vibration wave Sinusoidal logarithmic sweep. Sweep 0.5 octaves/min.	-Transportation environmental grade Level I II III IV -Use trucks for transportation.

Chapter 3

Work plan and procedure

3.1 Work plan

Table 3.1 Work plan

Subject	August				September				October			
Internship Orientation												
Product research												
Setup 3D CAD Training, SIM Training												
Design, Simulate												
Environment Test												
House Keeping(Report, Presentation)												

 Blue color means expected time period.

 Red color means used time period.

3.2 Work details

- Drawing 3D mechatronics for network product by using Creo Elements/Direct 17.0 Modeling.
- Mechanical Analysis by using ANSYS Workbench platform.
- Writing report and assembly manual for network product.
- Environment testing such as IP test and vibration test.

Learn a basic use of Creo Elements/Direct 17.0 Modeling program tools and practice by drawing basic 3D models such as a bracket, to be a basic knowledge to further drawing skills.

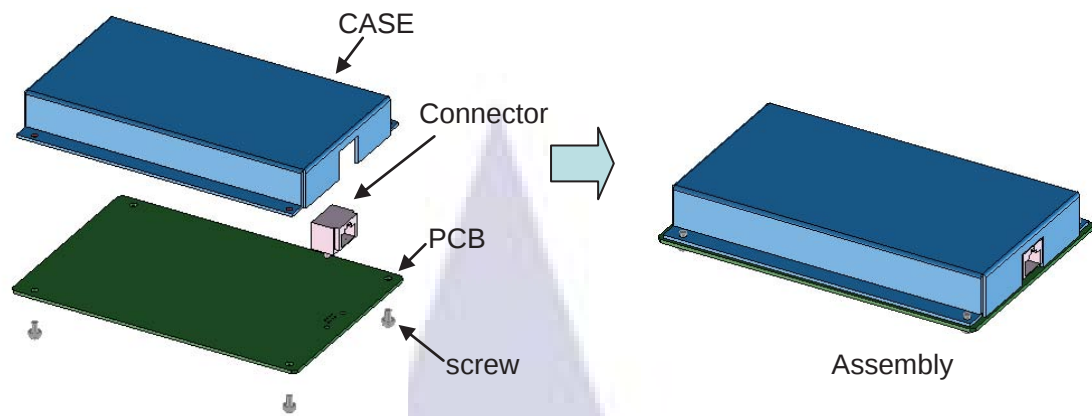


Figure 3.2 3D bracket by Creo Elements/Direct

ANSYS Workbench platform

Learn how to use ANSYS Workbench platform tools to simulate or analyze a 3D model from example case study for understanding of program basic use.

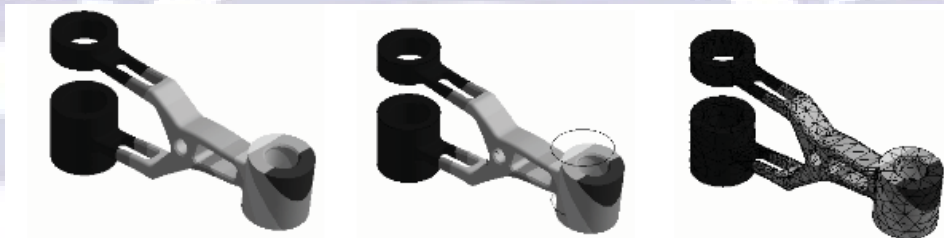


Figure 3.3 3D analysis by ANSYS Workbench platform

3.3.4 Design, Simulate

Designing and drawing 3D models from assigned case using Creo Elements/Direct 17.0 Modeling and analyze assigned 3D models from assigned case using ANSYS Workbench platform. Writing report and making manual for network product from assigned case study are also in this process.

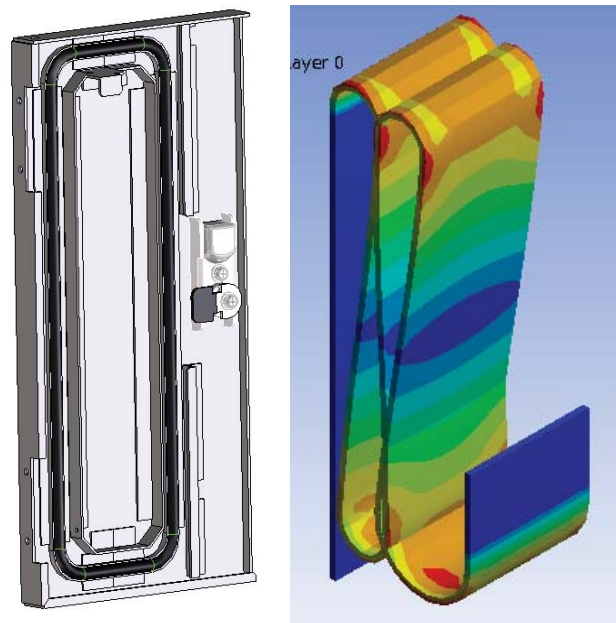


Figure 3.4 Design and Simulate 3D analysis

3.3.5 Environment Test

Randomly test sampling network products to check that product will perfectly protected in various environments and writing report of test results.

Environment test which I have done are as follows

IP test

IP test is the test to confirm the degrees of protection provided by the enclosures of electrical equipment. The form of the test will be in accordance with the standards set.



Figure 3.5 IP test process

Vibration test

Vibration test is the test to confirm that network product will perfectly safe during the delivery period. The form of the test will be in accordance with the standards set.



Figure 3.6 Vibration test process

3.3.6 House Keeping

Writing report and Making a presentation to sum up overall internship experience.

Chapter 4

Working Procedure and Analyze

4.1 Workflow Conclusions

During the internship period at the organization for 42 days, the workflow can be summarized as follows

- Talking about internship process.
- Basic learning of program use.
- Drawing 3D mechatronics for network product from assigned case studies using Creo Elements/Direct 17.0 Modeling.
- Mechanical Analysis from assigned case studies using ANSYS Workbench platform.
- Writing report and assembly manual for network product.
- Environment testing for network products.
- Writing report of environment test.
- Writing report and making presentation to sum up overall internship experience.

4.2 Analysis Results

After learn and practice from some case studies, results can be summarized as follows

4.2.1 Result of learning Creo Elements/Direct 17.0 Modeling.

I can draw 3D CAD models using Creo Elements/Direct 17.0 Modeling. To develop and design mechatronics for network product, I use steps of drawing as follows.

- Create workplane.

Create new workplane to be a base for building up 3D model.

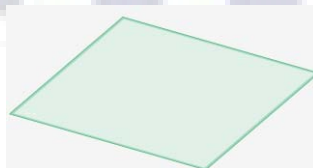


Figure 4.1 Create workplane.

- Draw guides.

Draw 2D guides on any workplane to be a layout for drawing profiles.

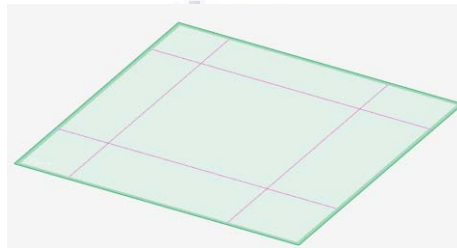


Figure 4.2 Draw guides.

- Draw a profile.

Draw a 2D profiles for building up a 3D object.

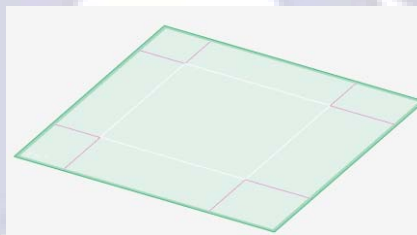


Figure 4.3 Draw profile

- Machine into 3D geometry.

Build up a 3D Object from a profile on any workplane.

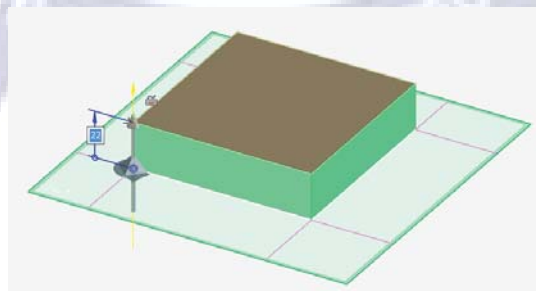


Figure 4.4 Machine into 3D geometry

- Modify geometry.

Modify 3D object such as blend a corner, punch an object.

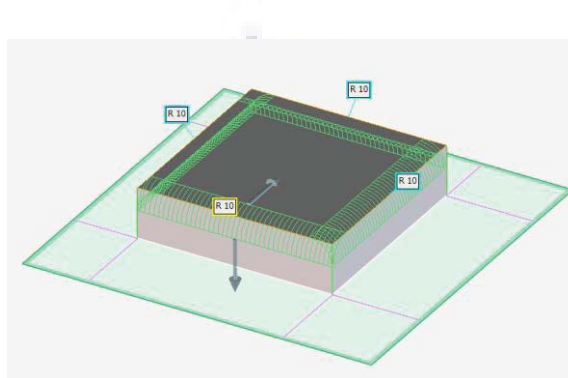


Figure 4.5 Modify geometry

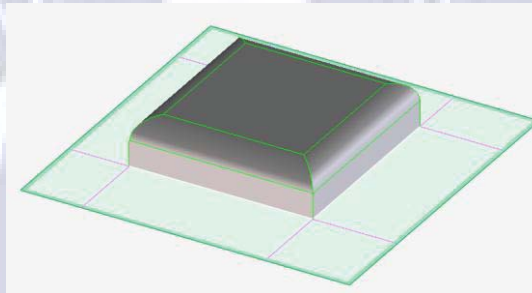


Figure 4.6 Result of Modify geometry

- Add new workplane to modify

Workplane can be create as many as user want.

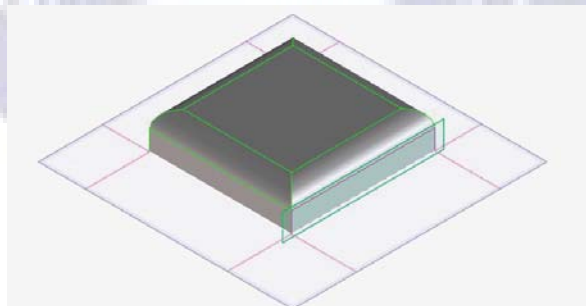


Figure 4.7 Add new workplane

- Draw guidelines and profile on new workplane.



Figure 4.8 Draw guidelines and profile on new workplane.

- Modify object

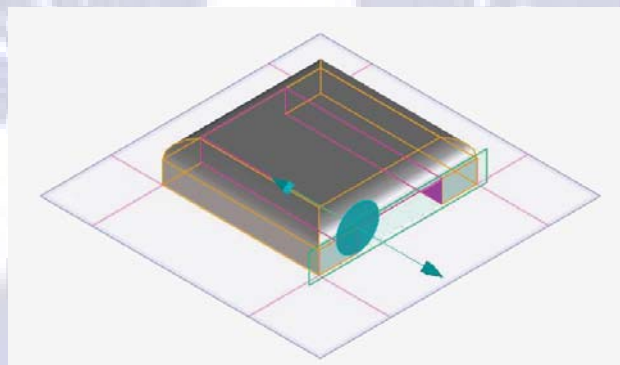


Figure 4.9 Modify object

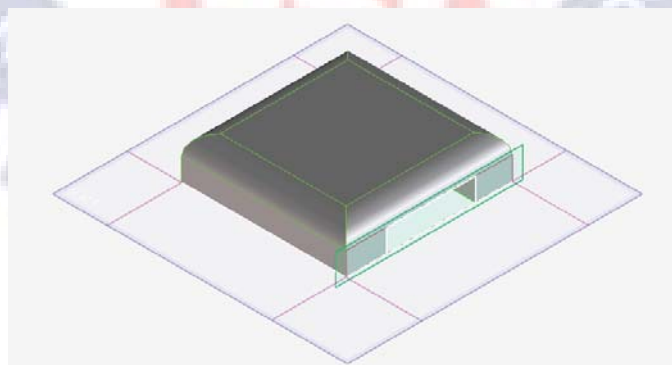


Figure 4.10 Result of Modify object

- Generate a drawing.

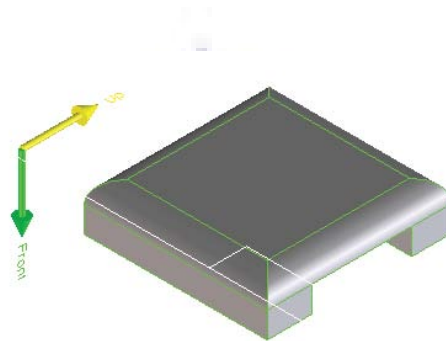


Figure 4.11 Select Object to generate a drawing

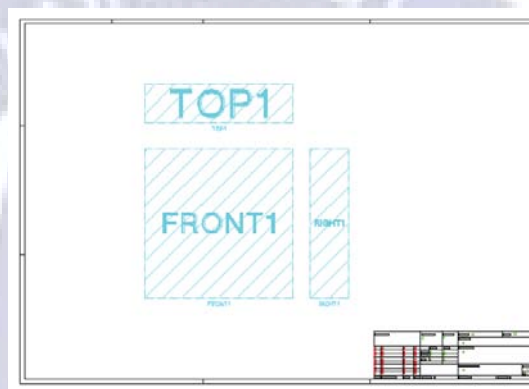


Figure 4.12 Select view to generate a drawing

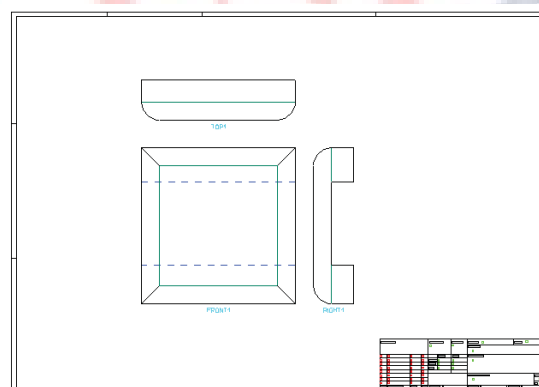


Figure 4.13 Generate a drawing

- Define details of drawing

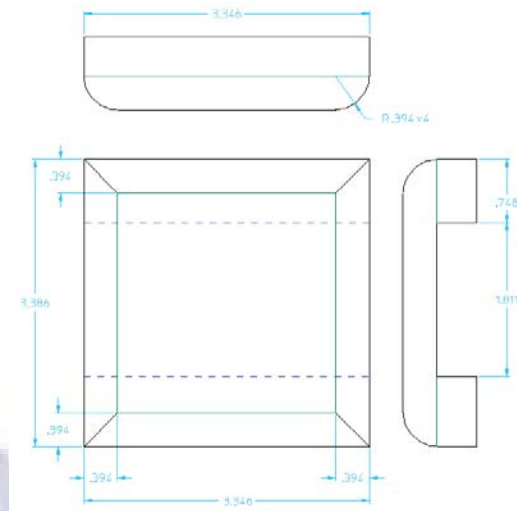


Figure 4.14 Define details of drawing

- Save file

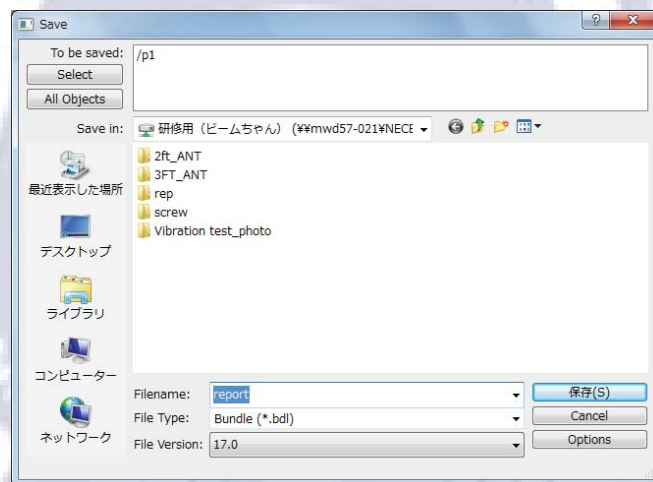


Figure 4.15 Save file

- Assembly

Objects can be assembled together by creating an assembly file.

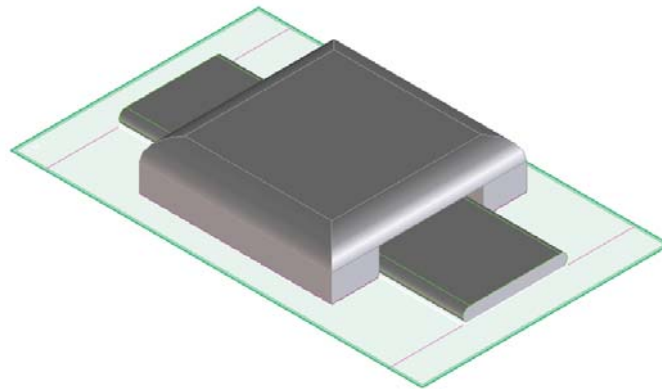


Figure 4.16 Assembled objects

- Position

Objects can be moved to any position.

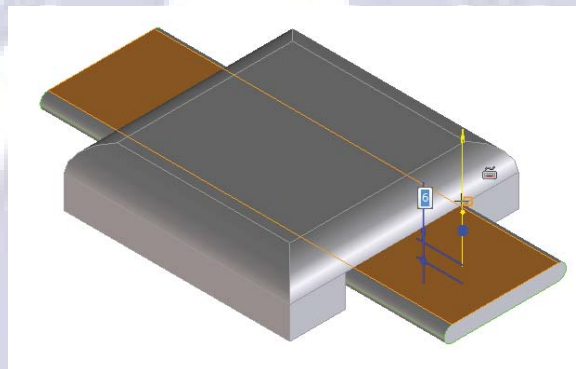


Figure 4.17 Position assembled objects

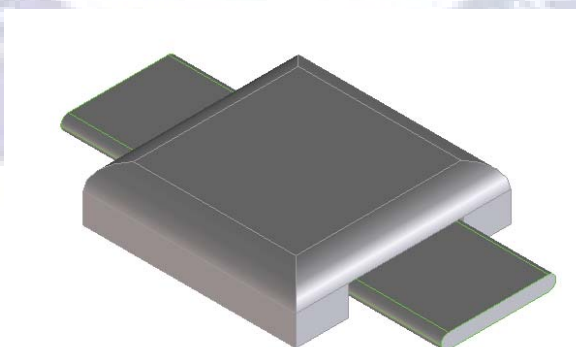


Figure 4.18 Result of positioning assembled objects

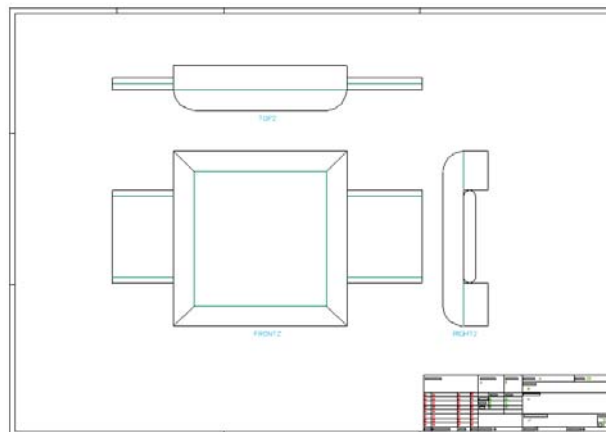


Figure 4.19 Generate drawing of assembled objects

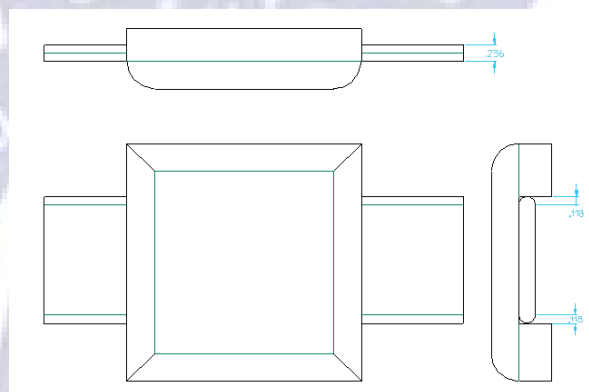


Figure 4.20 Define details of drawing of assembled objects

- Coloring an object and select material

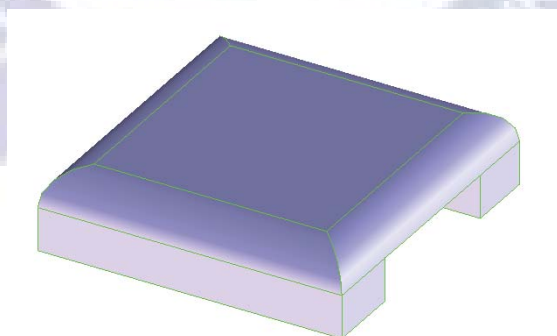


Figure 4.21 Coloring object



Figure 4.22 Select material for object



Figure 4.23 Example of assigned work using Creo Elements/Direct

4.2.2 Result of learning ANSYS Workbench platform

I can analyze 3D CAD models which is drawn by Creo Elements/Direct 17.0 Modeling by using ANSYS Workbench platform. To analyze mechatronics , I use steps of drawing as follows.



Figure 4.24 ANSYS Workbench platform start up

Simulate an analysis

- Activate ANSYS Workbench

Start->Allprograms->ANSYS14.0->Workbench

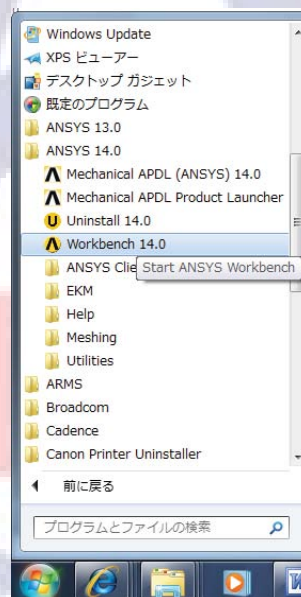


Figure 4.25 Activate ANSYS Workbench

- Import the geometry

Right-click the Geometry tab, select “Import Geometry” and Browse to select file.

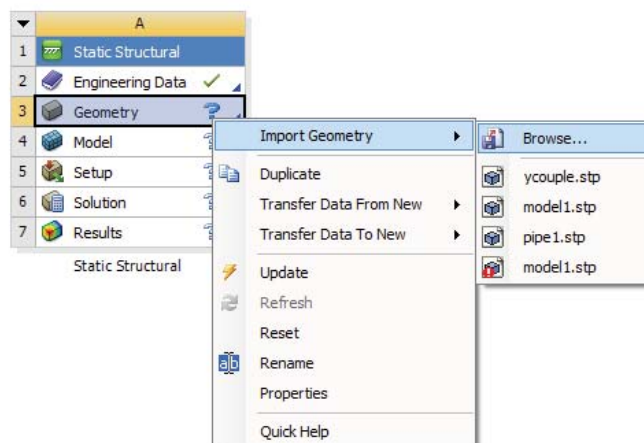


Figure 4.26 Import the geometry

In this example is YCOUPLE.prt file.

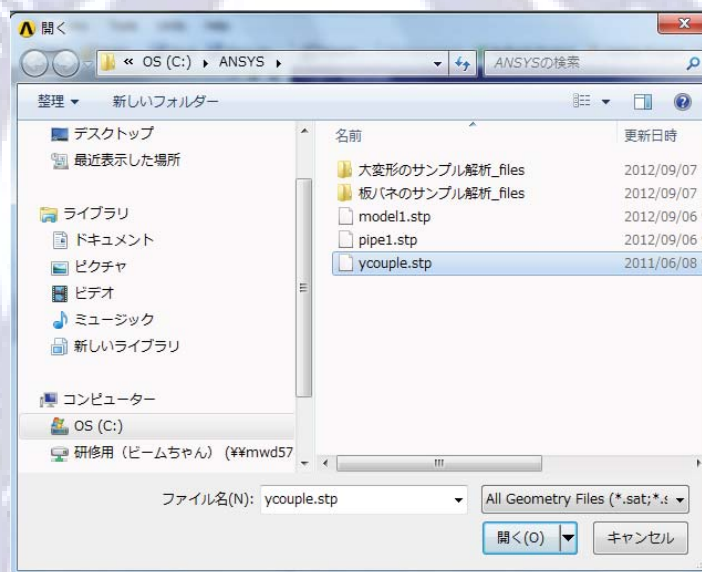


Figure 4.27 Select file to import the geometry

- Activate Mechanical

Double-click the Model tab to activate mechanical.

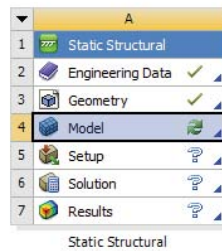


Figure 4.28 Activate Mechanical

- Confirm the material

Select “YCOUPLE.prt” in the Geometry in Outline tab.

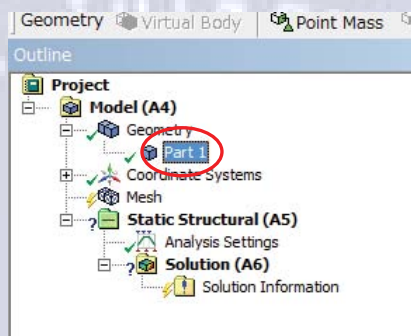


Figure 4.29 Select material to confirm

Make sure that the Assignment in the Material is Structural Steel

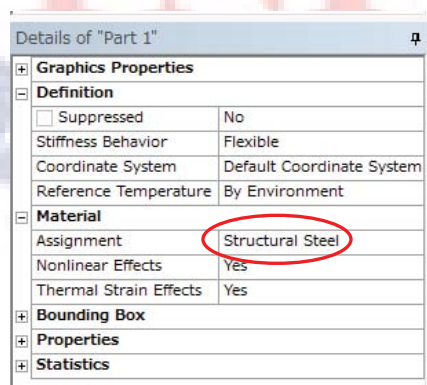


Figure 4.30 Confirm the material

Some case, material has to be added. Double-click the Engineering data tab.

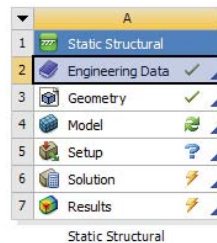


Figure 4.31 Click the Engineering data tab

Select Engineering Data Sources

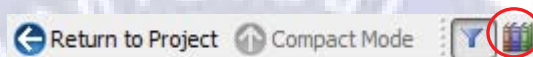


Figure 4.32 Select Engineering Data Sources

Select wanted data source, the data source outline will be shown.

Engineering Data Sources				
	A	B	C	D
	Data Source		Location	Description
1	★ Favorites			Quick access list and default items
2	一般材料			General use material samples for use in various analyses.
3	一般非线性材料			General use material samples for use in non-linear analyses.
4	瞬态材料			Material samples for use in an explicit analysis.
5	超弹性材料			Material stress-strain data samples for curve fitting.
Outline of 一般材料				
	A	B	C	D
	Contents of 一般材料		Add	source
				Description
1	Material			
2				
3	Structural Steel			Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
4	Air			General properties for air.
5	Aluminum Alloy			General aluminum alloy. Fatigue properties come from MIL-HDBK-5H, page 3-277.
6	Concrete			
7	Copper Alloy			
8	Gray Cast Iron			
9	Magnesium Alloy			
10	Polyethylene			
11	Stainless Steel			
12	Titanium Alloy			
13	Silicon Anisotropic			

Figure 4.33 Select wanted data source

In data source outline, select wanted source. In this case is Aluminum Alloy.

	A	B	C	D	E
1	Contents of 材料		Add	Source	Description
2	Material				
3	Structural Steel				Fatigue Data at zero mean stress comes from 1988 ASME BPV Code, Section II, Div 2, Table 5-110.1
4	Air				General properties for air.
5	Aluminum Alloy				General aluminum alloy. Fatigue properties come from MIL-HDBK-5H, page 5-277.
6	Concrete				
7	Copper Alloy				
8	Gray Cast Iron				
9	Magnesium Alloy				
10	Polycarbonate				
11	Stainless Steel				
12	Titanium Alloy				
13	Silicon Anisotropic				

Figure 4.34 Select wanted source

Confirm Engineering Data Sources



Figure 4.35 Confirm Engineering Data Sources

Return to Project



Figure 4.36 Return to Project

- Set up Unit

Select “Units” in main menu bar and select “Metric(mm, kg, N, s, mV, mA)”.

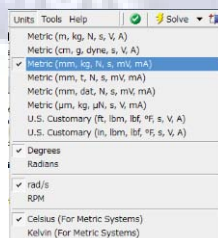


Figure 4.37 Set up Unit

- Define force on object

Select “Static Structural” in Outline tab.

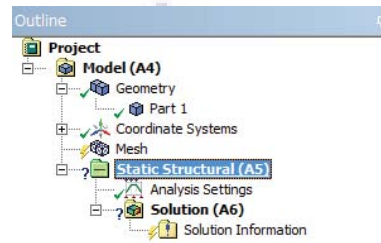


Figure 4.38 Define force on object

Select “Loads” on the context toolbar and click “Force”.

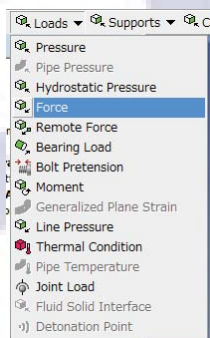


Figure 4.39 Click “Force”

Select the inner face (green) of cylinder hollow to define force on it.

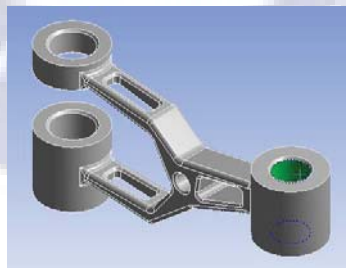


Figure 4.40 Select the inner face to define force

In details of Force, click “Apply” to confirm.

Details of "Force"	
Scope	
Scoping Method	Geometry Selection
Geometry	Apply Cancel
Definition	
Type	Force
Define By	Vector
☒ Magnitude	0. N (ramped)
Direction	Click to Define
Suppressed	No

Figure 4.41 Confirm force

At the Define By tab, change from “Vector” to “Components”.

Details of "Force"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Force
Define By	Vector
☒ Magnitude	Components
Direction	Vector
Suppressed	No

Figure 4.42 Change Define by

At the X Component tab, fill a value of “1000”.

Details of "Force"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
☒ X Component	1000
☐ Y Component	0. N (ramped)
☐ Z Component	0. N (ramped)
Suppressed	No

Figure 4.43 Change value of Coordinate System

In the geometry window, make sure that the force direction is +X

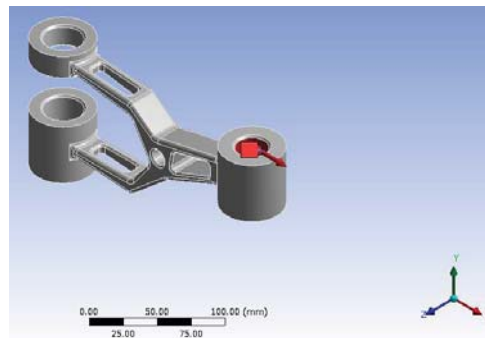


Figure 4.44 Confirm the force direction

- Define Fix part on object

Select “Supports” on the context toolbar and select “Fixed Support”.

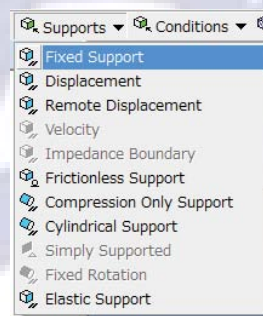


Figure 4.45 Define Fix part on object

Hold down the Ctrl Key while select 2 faces(green) as picture.

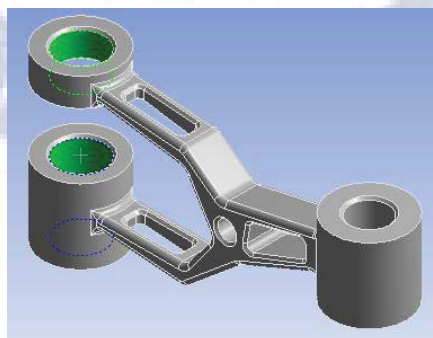


Figure 4.46 Select 2 faces to fix

In details of “Fixed Support”, click “Apply” to confirm.

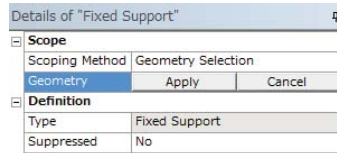


Figure 4.47 Confirm Fix part on object

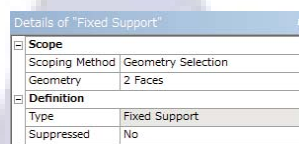


Figure 4.48 Fix part on object is confirmed

2. Display the solution of simulation.

- Select type of solution to display
- Select “Solution” in Outline tab.

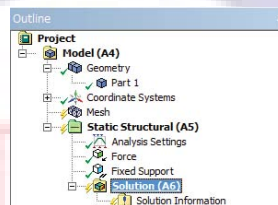


Figure 4.49 Select Solution

Select “Stress” on the context toolbar and select “Equivalent(von-Mises)”

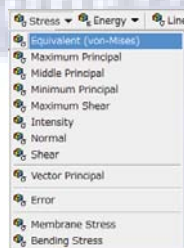


Figure 4.50 Select Equivalent(von-Mises)

Select “Stress” on the context toolbar and select “Vector Principal”

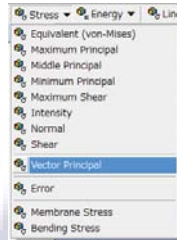


Figure 4.51 Select Vector Principal

Select “Deformation” on the context toolbar and select “Total”

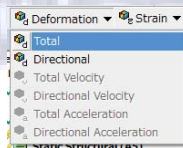


Figure 4.52 Select Total

- Solve

Click “Solve” in the standard toolbar.

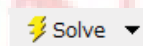


Figure 4.53 Solve button

- Analysis

Wait for analysis progress.

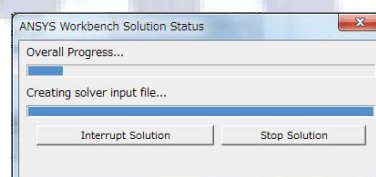


Figure 4.54 Analysis progress

After analysis is finish, select “Equivalent Stress” in the solution in Outline tab.

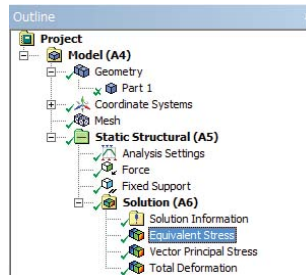


Figure 4.55 Select Equivalent Stress

The contour figure is displayed in the geometry window.

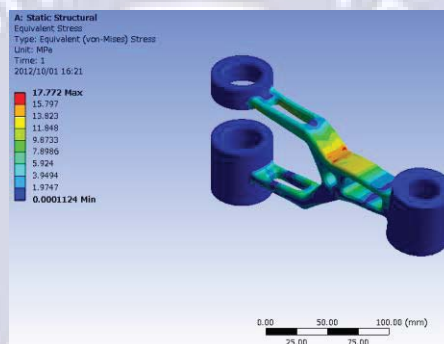


Figure 4.56 Contour figure is displayed

See results of Minimum and Maximum.

Details of "Equivalent Stress"	
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Equivalent (von-Mises) St...
By	Time
Display Time	Last
Calculate Time History	Yes
Identifier	
Suppressed	No
Integration Point Results	
Display Option	Averaged
Results	
Minimum	1.124e-004 MPa
Maximum	17.772 MPa
Information	

Figure 4.57 Result of Minimum and Maximum

- Section

Click “New section plane” in the standard toolbar.

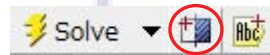


Figure 4.58 New section plane button

Define the direction of the section. In geometry window, right-click->View->Top.

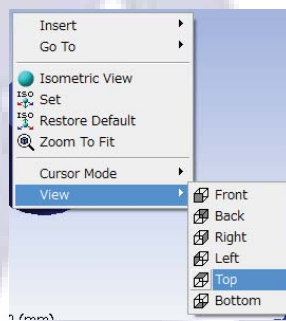


Figure 4.59 Define the direction of the section

Drag mouse in wanted direction, the section will be created.

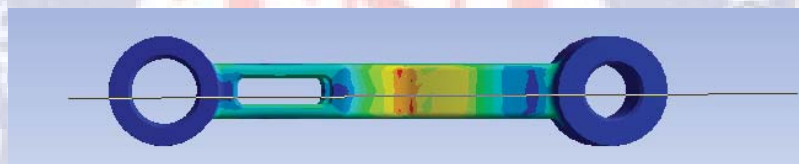


Figure 4.60 The section is created

Click “Rotate” in toolbar and rotate the geometry to see the section.

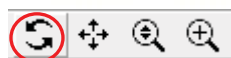


Figure 4.61 Rotate button

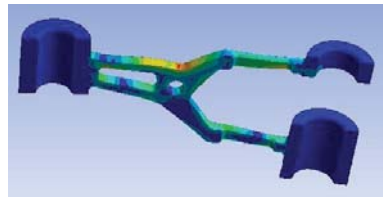


Figure 4.62 Rotate the geometry

Select the slice Plane 1 in the section view to display the anchor in the geometry window.



Figure 4.63 Select the slice Plane

Section plane can be move by dragging the center box of anchor.

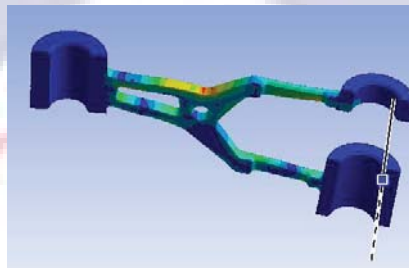


Figure 4.64 Move section plane

To release the display of section, uncheck a checkbox in the section view. If there are multiple sections, using following menu to modify each section.

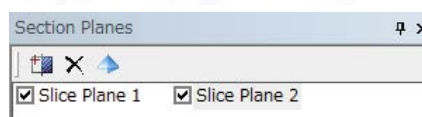


Figure 4.65 Select the slice Planes

-  Create new section
-  Delete a section
-  Display all elements

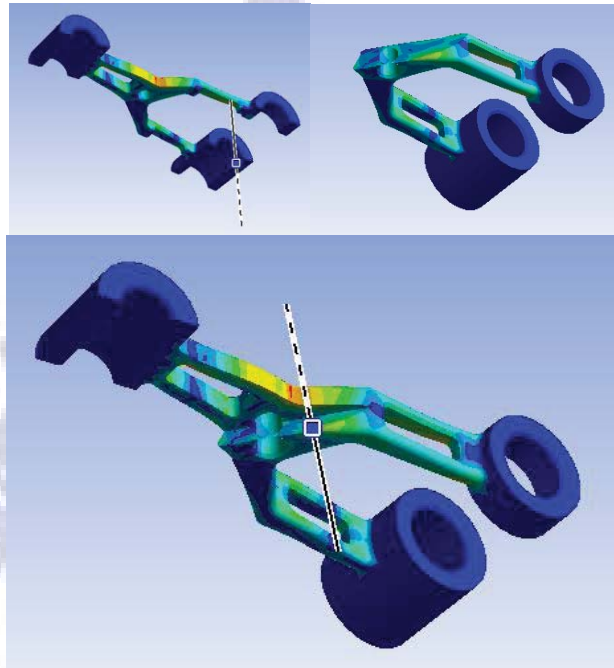


Figure 4.66 Result of modifying slice plane

- Display Vector Principal Stress

Select “Vector Principal Stress” in the solution in Outline tab, the vector figure of principal stress will be displayed on the geometry window.

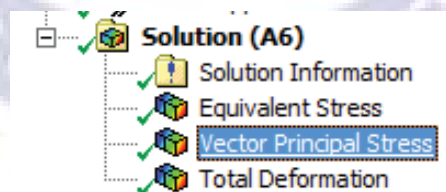


Figure 4.67 Select Vector Principal Stress

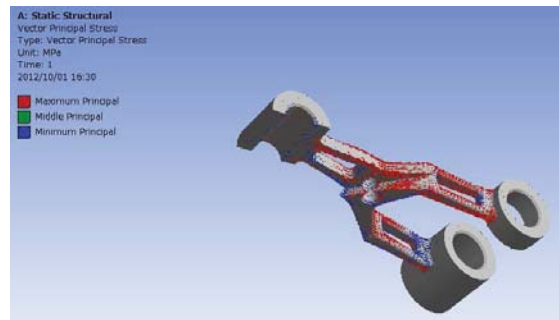


Figure 4.68 Vector figure of principal stress is displayed

- Display total deformation

Select “Total Deformation” in the solution in Outline tab, the contour figure of total deformation will display on the geometry window.

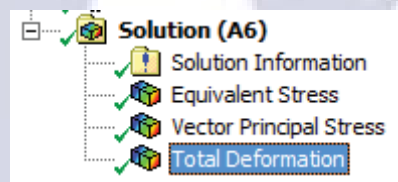


Figure 4.69 Select Total deformation

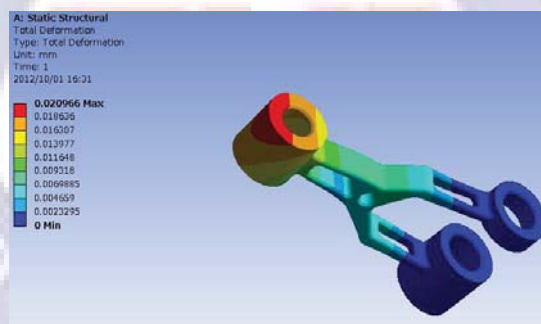


Figure 4.70 Contour figure of total deformation is displayed

Select “Edges” on the context toolbar and select “Show Undeformed Model”, the translucent undeformed figure will display overlapping with deformed part on the geometry window.

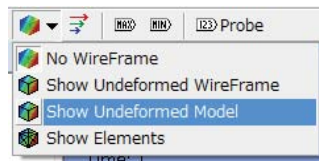


Figure 4.71 Select Show Undeformed Model

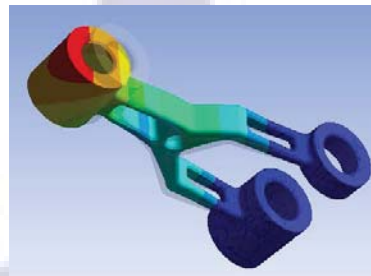


Figure 4.72 Undeformed Model is displayed

- Display in animation

Solution can be displayed with an animation by, Display the contour which is wanted to animate.



Figure 4.73 Select contour

Select “Equivalent Stress” in the solution on Outline tab.

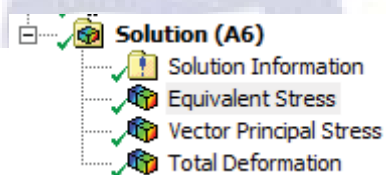


Figure 4.74 Select Equivalent Stress

Click the play button in the graph view, the animation will be played.



Figure 4.75 Click the play button

3. Create report

- Insert figures

Figures can be inserted to report by,

Select “Equivalent Stress” in the solution in Outline tab.

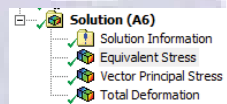


Figure 4.76 Select type of solution

Select “New Figure or Image” in the standard toolbar, and click “Figure”, figured will be inserted.

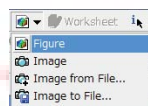


Figure 4.77 Figure is inserted.

Figures can be inserted anywhere in model.

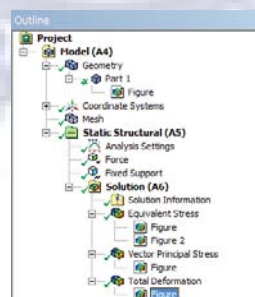


Figure 4.78 Figure is inserted anywhere user wanted

Click “Report Preview” in the document tab..

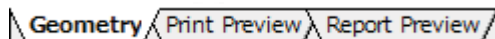


Figure 4.79 Select Report Preview

Wait for creating the report is finished, Report will be displayed.



Report generation in progress...

Please wait while the system extracts all necessary project information.
During this process, please refrain from all project interaction.

Figure 4.80 Report in progress

Save the report.

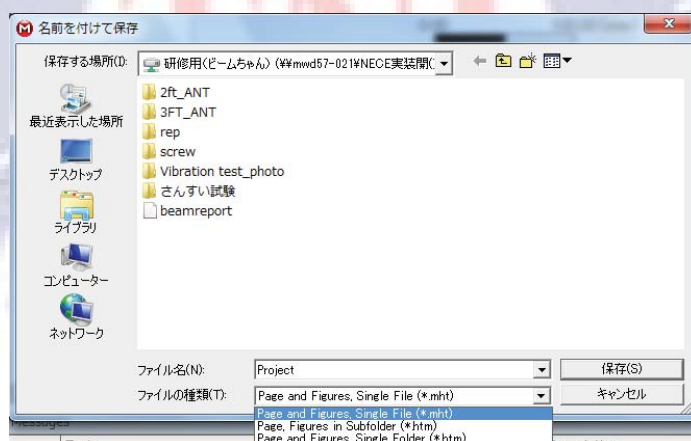


Figure 4.81 Save the report

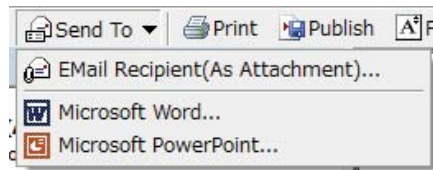


Figure 4.82 Send report to other program

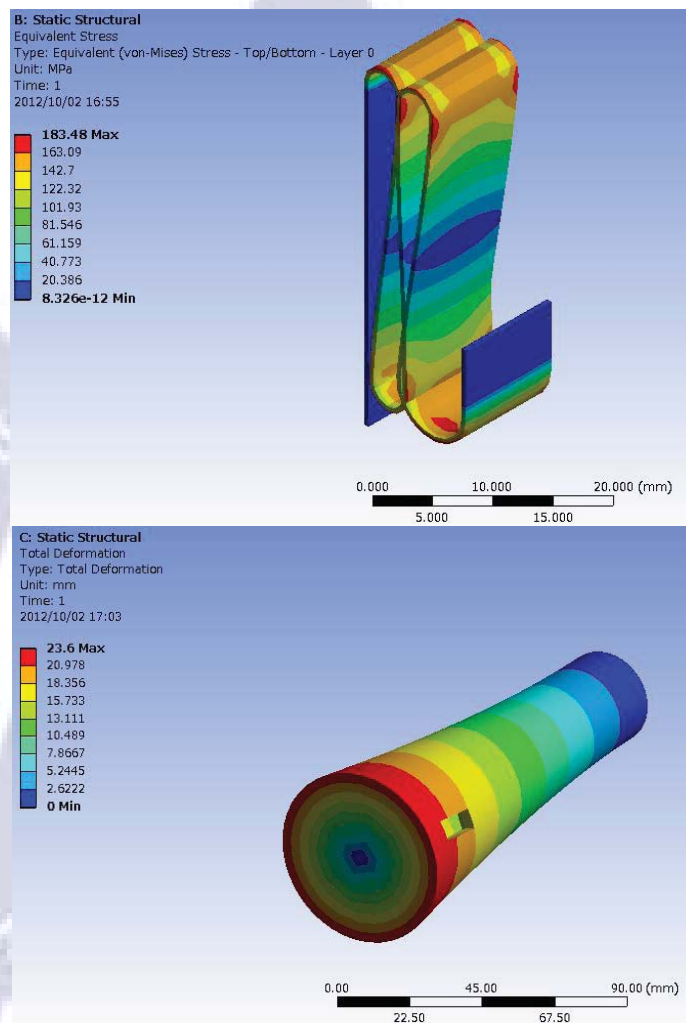


Figure 4.83 Example of assigned work using ANSYS Workbench platform

4.2.3 Result of writing and making network product installation manual

Learn to assembly network products for writing and making installation manual.

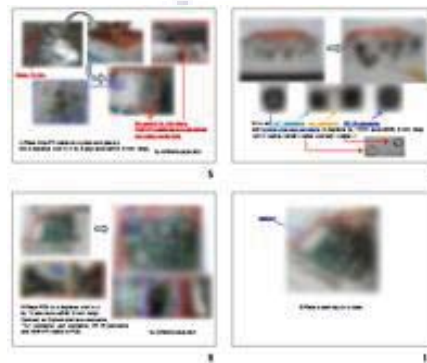


Figure 4.84 Example of assigned work of writing and making installation manual

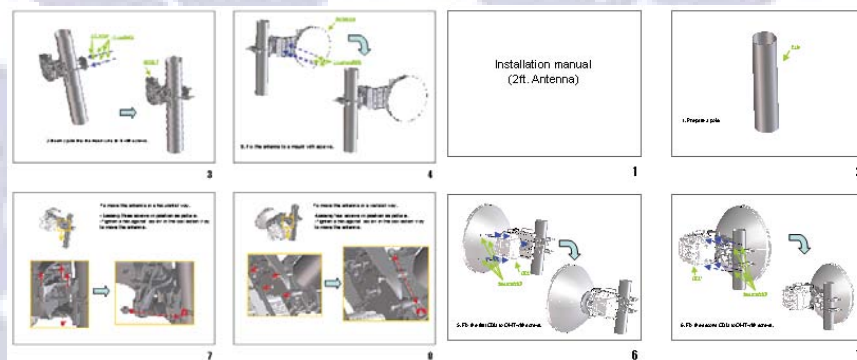


Figure 4.85 Example of assigned work of writing and making installation manual using Creo Elements/Direct

4.2.4 Result of learning about environment testing

IP test

IP test is the test to confirm the degrees of protection provided by the enclosures of electrical equipment.

- Prepare

Everything has to be set into its standard. For example, you have to use 12.5 mm nozzle if you want to test an IPx6 and have to use 6.3 mm nozzle if you want to test IPx5.

- Test

As above, the test method is up to standard of IP code. For example, if protection against ingress of water (IP code) is IPx6. The test will be conducted with fresh water and will be made by spraying the test item with a stream of water from one standard nozzle. The water jet hose nozzle will be held in a distance of about 2.5 from test item during the test. Two part of the test item will be tested for a total time of 6 min. (3 minute each session), adjusting the water delivery rate to 100 liter per minute at the pressure of 0.15 m/s.



Figure 4.86 Example of assigned work of IP test

- Writing report

Report will be written to conclude a result of test. Important information is required such as standard number.



Figure 4.87 Example of report of IP test

Vibration Test

Vibration test is various used. In this case, I will show a information of vibration test for transportation.

- Prepare

Install an enclosure on the vibration plate of the tester to prevent the sample from being shaken off the plate. Place the sample in the enclosure area in a free state without fixing it in a position. Then, Place the sample fixed on the vibration plate.

- Test

As above, the test method is up to standard of JIS Z 0200 and JIS Z 0232. For example, test specifications are as follows

1. Vibration direction: x, y, z
2. Frequency range: 3 Hz – 100 Hz
3. The amplitude and Vibration acceleration: 3-4 Hz: 25 mm(p-p) , 4-100 Hz: ± 0.75 G
4. Test period: 3 hours (60 minutes per each direction)



Figure 4.88 Example of assigned work of Vibration test

Product has to be tested in 3 directions and take 3 hours (1 hour each). After testing and checking for result, Heat sink should not get free from part of product and screws should not loose. Also, the product and part of product should not be ruined.



Figure 4.89 Example of checking result of Vibration test

- Writing report

Report will be written to conclude a result of test.

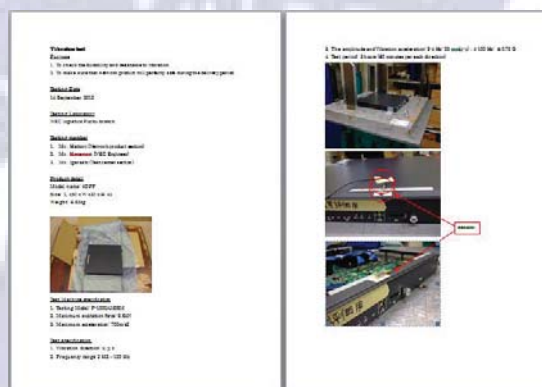


Figure 4.90 Example of report of Vibration test

4.3 Comparing result of work with working purpose

From comparing the result of work process with work purpose, I can draw 3D models using Creo Elements/Direct 17.0 Modeling from any assigned work. I can draw a mimic object by reading drawing sheets or measuring from real objects, which can assume the general shape of any object. I can also analyze 3D model by using ANSYS Workbench platform to test network products in many conditions. Moreover, I have learned about environment test and understand about its procedure, purpose and result checking.

From above mentioned, I can learn, practice and further my knowledge and skills as expected.

Chapter 5

Conclusions and Suggestions

5.1 Conclusions

As working during internship period of 6 weeks, the process of mechatronics development and design for network product begin with researching and comparing properties of various network products. I have learned to design and draw 3D models by Creo Elements/Direct 17.0 Modeling and analysis by ANSYS Workbench platform. I can draw and analyze 3D models in many case studies and assigned works. I also did environment testing (IP test and Vibration test) to test sampling product in various case of environment and have a chance to write result reports.

5.2 Work Conclusions

From above mentioned, I can learn, practice and further my knowledge and skills as follows

Creo Elements/Direct 17.0 Modeling

After basic tutorial, I can apply my skills of using tools to draw up both basic 3D models and advance 3D models.

ANSYS Workbench platform

After basic tutorial, I understand the purpose of using this program and can use basic tools to analyze 3D models in case studies.

Environment test

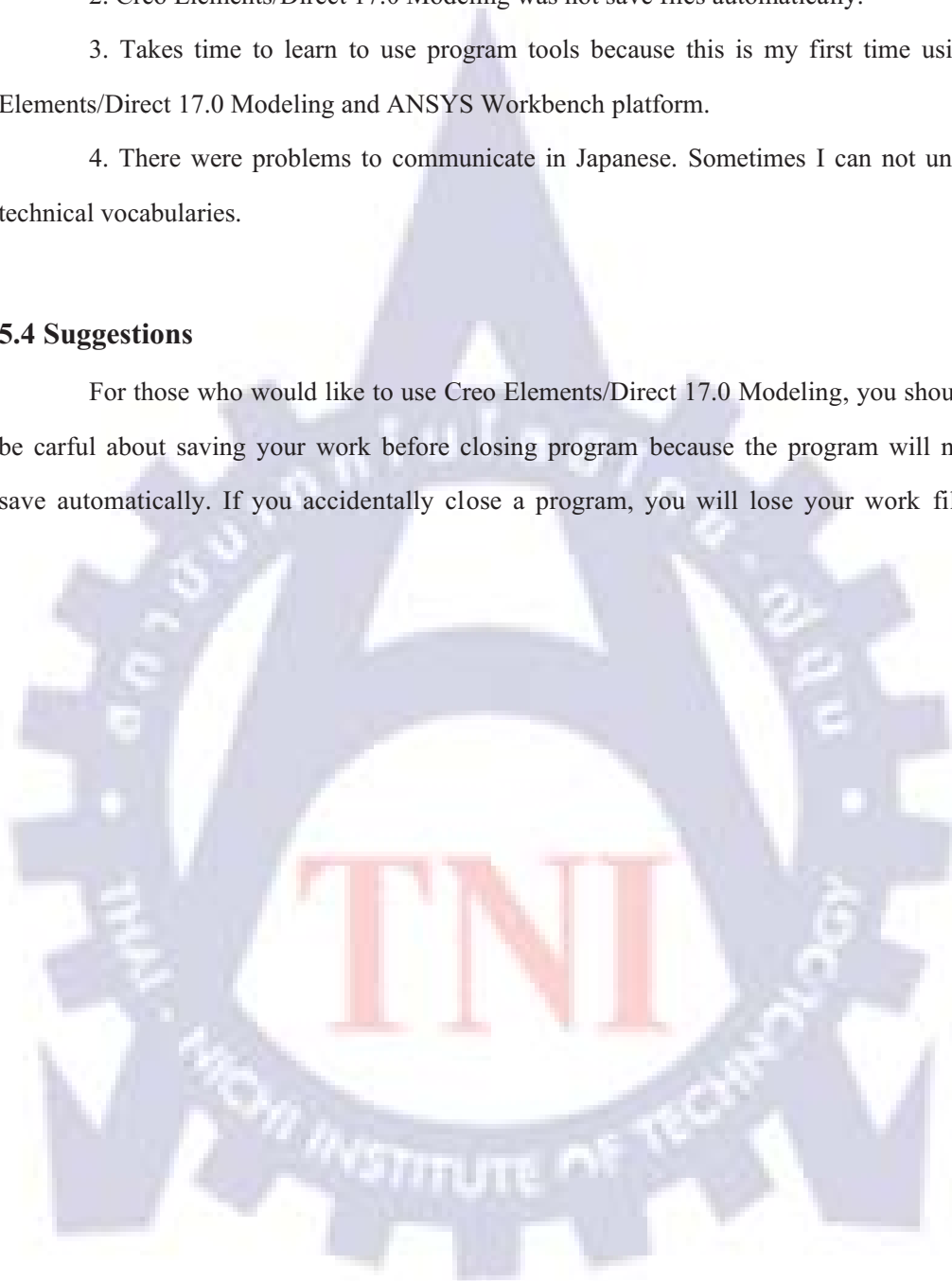
After go out for learning about environment test, I understand the purpose and standard of test. I can also make a conclusion based on test results.

5.3 Problems

1. Program processing was slow.
2. Creo Elements/Direct 17.0 Modeling was not save files automatically.
3. Takes time to learn to use program tools because this is my first time using Creo Elements/Direct 17.0 Modeling and ANSYS Workbench platform.
4. There were problems to communicate in Japanese. Sometimes I can not understand technical vocabularies.

5.4 Suggestions

For those who would like to use Creo Elements/Direct 17.0 Modeling, you should be careful about saving your work before closing program because the program will not save automatically. If you accidentally close a program, you will lose your work file.



References

1. NEC Corporation, 2012, **NEC Corporation** [Online], Available: <http://www.nec.com/> [September 25, 2012]
2. Parametric Technology Corporation, 2012, **Creo Elements/Direct Modeling** [Online], Available: http://www.ptc.com/WCMS/files/99952/ja/4826B_CC_Proof_Changes_TS_ja.pdf [September 26, 2012]
3. Parametric Technology Corporation, 2012, **PTC Creo Elements/Direct Modeling** [Online], Available: <http://www.ptc.com/product/creo-elements-direct/modeling> [September 26, 2012]
4. Wikimedia Foundation, 2012, **Ansys** [Online], Available: <http://en.wikipedia.org/wiki/Ansys> [September 26, 2012]
5. FIGES A.S., 2012, **What is ANSYS** [Online], Available: <http://www.figes.com.tr/english/ansys/ansys.php> [September 26, 2012]
6. ANSYS, 2012, **ANSYS Workbench Platform** [Online], Available: <http://www.ansys.com/Products/Workflow+Technology/ANSYS+Workbench+Platform> [September 26, 2012]
7. Wikimedia Foundation, 2012, **IP Code** [Online], Available: http://en.wikipedia.org/wiki/IP_Code [September 28, 2012]
8. Wikimedia Foundation, 2012, **Vibration** [Online], Available: <http://en.wikipedia.org/wiki/Vibration> [September 28, 2012]

9. Wikimedia Foundation, 2012, **Vibration** [Online], Available:
<http://en.wikipedia.org/wiki/Vibration> [October 1,2012]
10. Engineering ToolBox, 2012, **Revision of JIS Z 0232: Packaged freight-Method of vibration test** [Online], Available: <http://www.engineeringtoolbox.com/index.html> [October 2, 2012]



The logo of TNI (Thaksin National Institute) is a large, light blue watermark in the background. It features a stylized 'A' shape with a gear-like border. Inside the 'A', the letters 'TNI' are written in red. The Thai text 'วิทยาลัยเทคโนโลยีอาชีวศึกษา' is written along the top curve of the gear, and 'TNI - THAI HIGH INSTITUTE OF TECHNOLOGY' is written along the bottom curve.

Appendix A
Program manual

Program manual

Creo Elements/Direct 17.0 Modeling

1. Create and modify Workplane

Create and modify workplane by using tool on the right-side menu bar.

Create workplane

Click Workplane or WP set and choose a form of workplane. For example,

- New : create new workplane.
- New + C. Geo: create new workplane with base guides.
- On Face: create new workplane on specified face.
- Project Geo: create new workplane with profiles which projected from object.
- Project Constr: create new workplane with guides which projected from object.

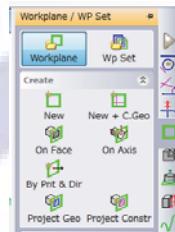


Figure A.1 Workplane tool

Create more

Workplane can also be created by depending on another workplane such as copying, share or parallel. Click Workplane or WP set and choose a tool from Create More.

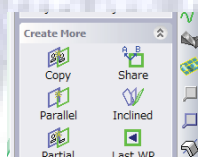


Figure A.2 Create more Workplane tool

Modify

Workplane can be modified in many ways such as moving to specified position.

Click Workplane or WP set and choose a tool from Modify or Modify More.

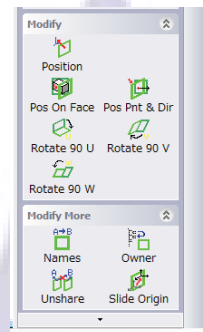


Figure A.3 Modify Workplane tool

2. Draw guides and profiles

Creating 2D

Draw guides and profiles by using 2D geometry toolbar or tools on the right-side menu bar. As the picture, Geometry will be selected to draw profiles. On the other hand, Construct will be selected to draw guides.

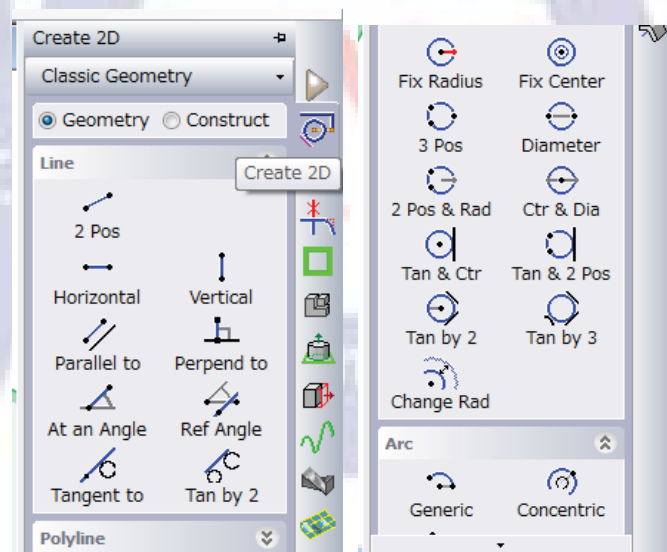


Figure A.4 Create 2D tool

Creo Elements/Direct 17.0 Modeling has many useful tools and features that will help drawing guides and profiles easy. For example, 3D Edge tool helps crating guide or profile from selected 3D edge on object. In addition, 2D edge on any workplane can be copied by 2D Edge tool.

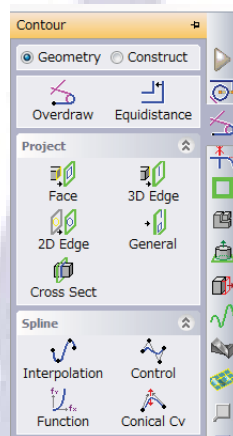


Figure A.5 Contour tool

Modify 2D

Guides and profiles can be modified in many ways such as filling fillets, moving or rotating lines and cut unwanted part of line.

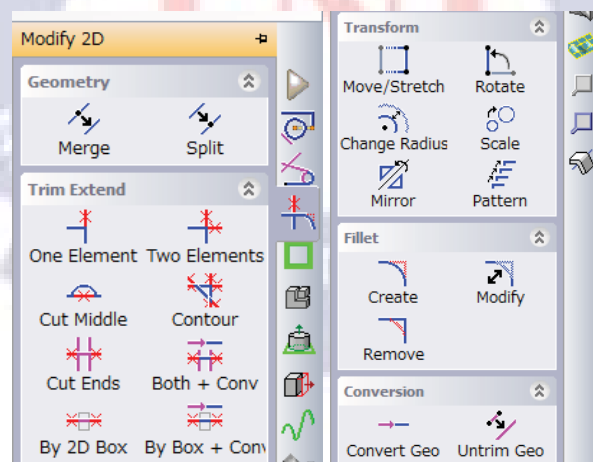


Figure A.6 Modify 2D tool

Guides and profiles can also be drawn or modified by 2D Geometry tool, there are list of commonly used tool such as

-  Draw a profile line.
-  Draw a profile rectangle.
-  Draw a profile circle.
-  Draw a guide line.
-  Draw a guide line which parallel to specified line.
-  Move 2D line.

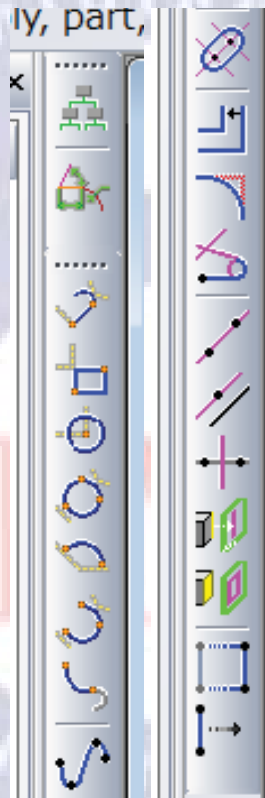


Figure A.7 2D Geometry tool

3. Machine into 3D

Machining 3D object from profiles on any workplane by using machine tools on the right-side menu bar, such as

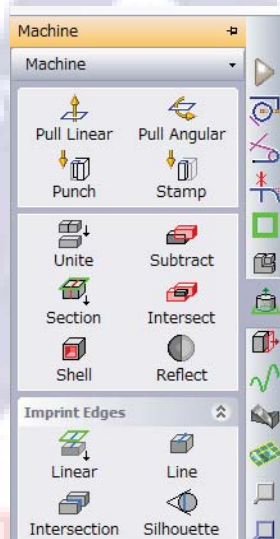
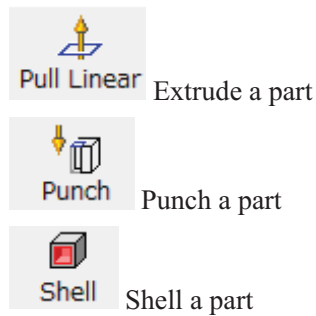


Figure A.8 Machine tool

The active part and workplane are selected by default. When creating new or work on different part is wanted, part and/or workplane have to be changed when machining command is started, if necessary.

Any changes are not complete until OK is clicked. Although the machining operation can be seen as a preview in the viewport, the changes are not saved unless the green checkmark is clicked.

All profiles on a workplane are machined. If the same workplane is used for more than one operation, unwanted profiles are needed to delete before the next operation begins.

4. Modify 3D

Modifying 3D object by using modify tools on the right-side menu bar, such as

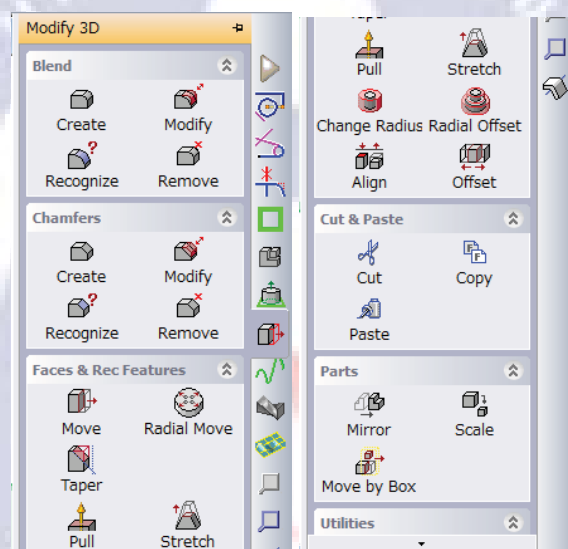
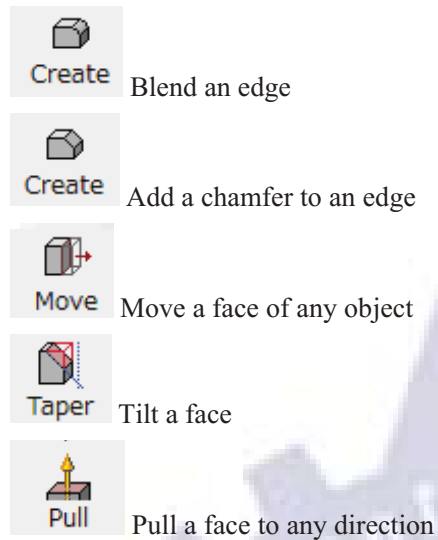
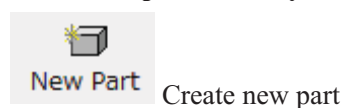


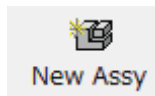
Figure A.9 Modify 3D tool

5. Assembly

Assembly parts together, using Part & Assy tool on the right-side menu bar.

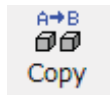
Creat new part and assy





New Assy

Create new assembly. In the Structure Browser, drag and drop the parts to be the contents as part of the assembly



Copy

Create new part by copying another.



Linear

Create many new parts by copying or sharing from another. Result of this tool is row of parts which have a distance between each as specified.

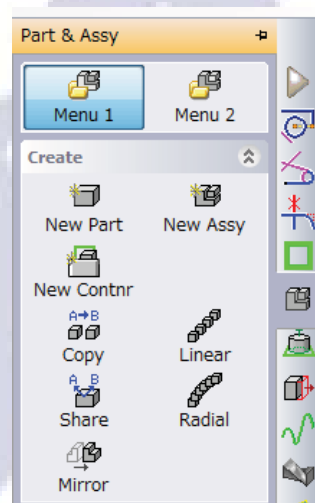
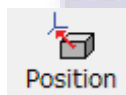


Figure A.10 Assembly tool

Modify part and assembly

Modify part and assembly, using Modify tool from Part & Assy tab on the right-side menu bar, such as



Position

Move specified part to specified position. This tool has many mode such as

Mate: Mates the faces, edges, or vertices together.

Align: Aligns two faces, edges, or vertices along the same plane.

Align Axis: Aligns the specified axes.

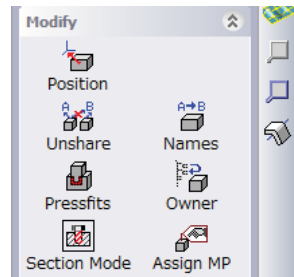


Figure A.11 Modify part and assembly tool

8. Drawing

Create drawing by select Application -> Modules and Select Annotation

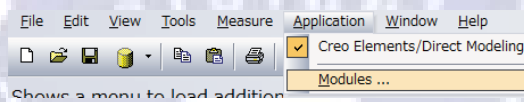


Figure A.12 Select Application

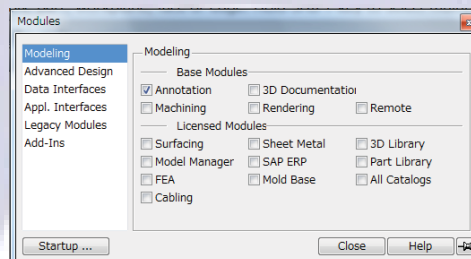


Figure A.13 Select Annotation

Create a drawing and view

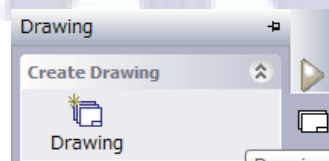


Figure A.14 Create Drawing tool

If necessary, change the default parameters, under Drawing:

Owner- the owner of the view.

Front Dir- the front direction of the view.

Up Dir- the up direction of the front view.

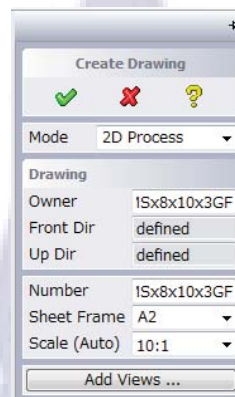


Figure A.15 Create Drawing parameters

Select Add Views to add views.

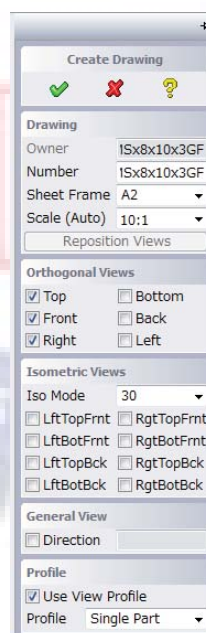


Figure A.16 Select Add Views to add views

9. Define material of objects, light and background

Create drawing by select Application -> Modules and Select Rendering

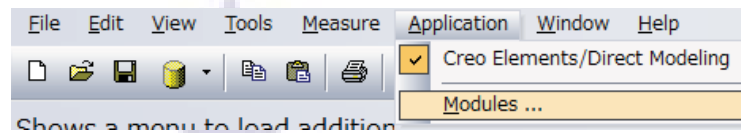


Figure A.17 Select Application

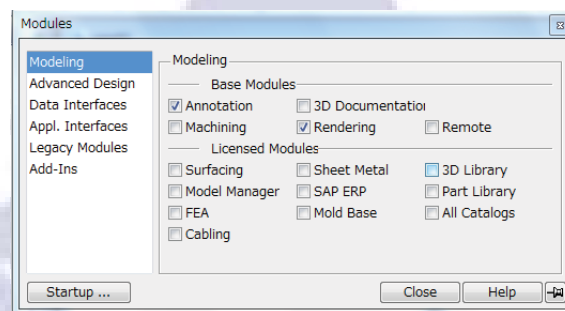


Figure A.18 Select Rendering

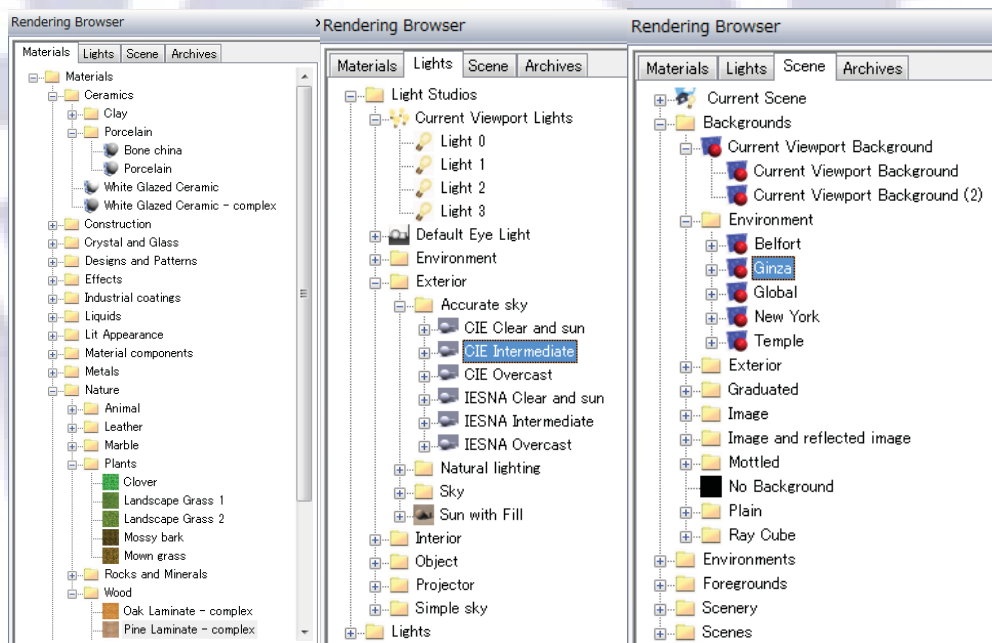


Figure A.19 Rendering Browser Windows

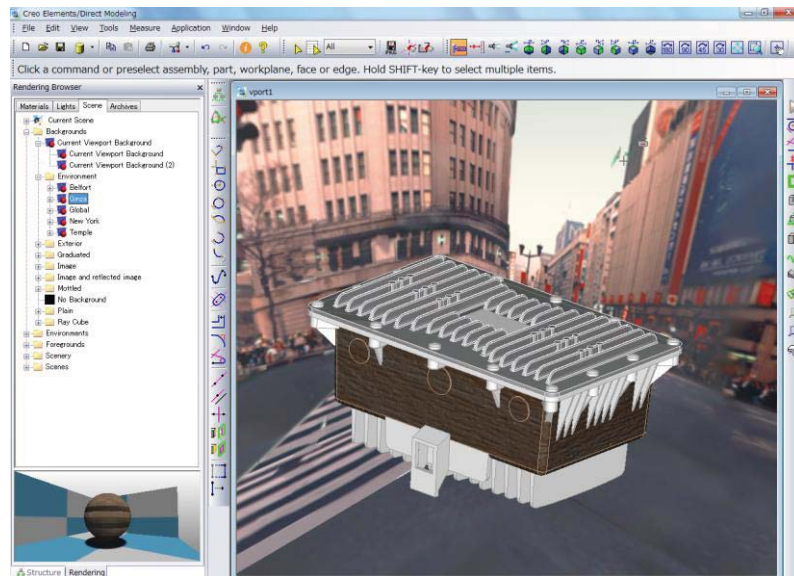
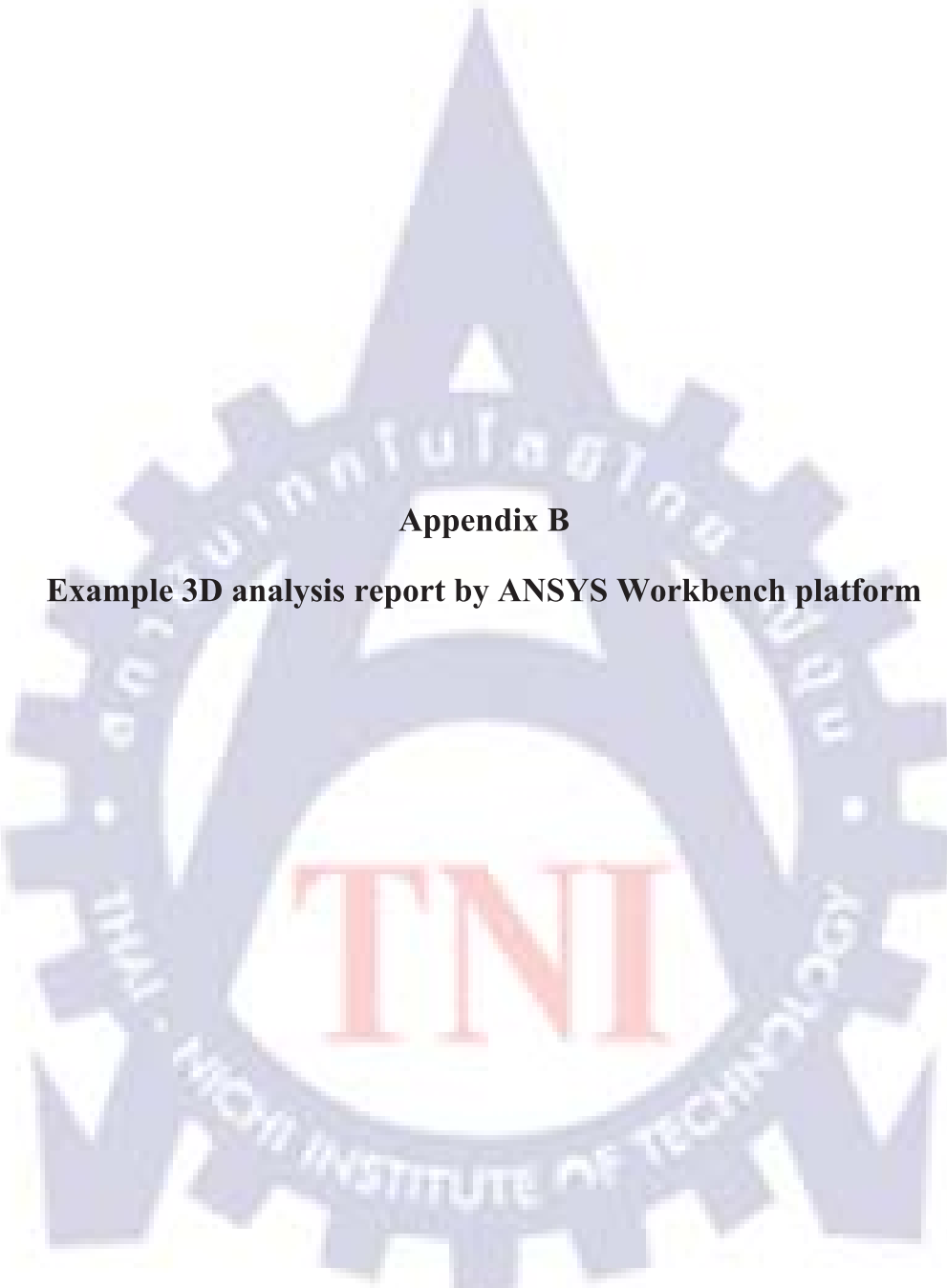


Figure A.20 Result of Rendering

The logo of TNI (Tribhuvan University) is a large, light blue watermark in the background. It features a gear-like outer ring with the text 'TRIBHUVAN UNIVERSITY' at the top and 'HIGHER INSTITUTE OF TECHNOLOGY' at the bottom. In the center is a stylized 'A' shape with a triangle inside, and the letters 'TNI' in red are prominently displayed in the middle.

Appendix B

Example 3D analysis report by ANSYS Workbench platform

Example 3D analysis report by ANSYS Workbench platform

Table B.1 Project

First Saved	Monday, October 01, 2012
Last Saved	Monday, October 01, 2012
Product Version	14.0 Release
Save Project Before Solution	No
Save Project After Solution	No

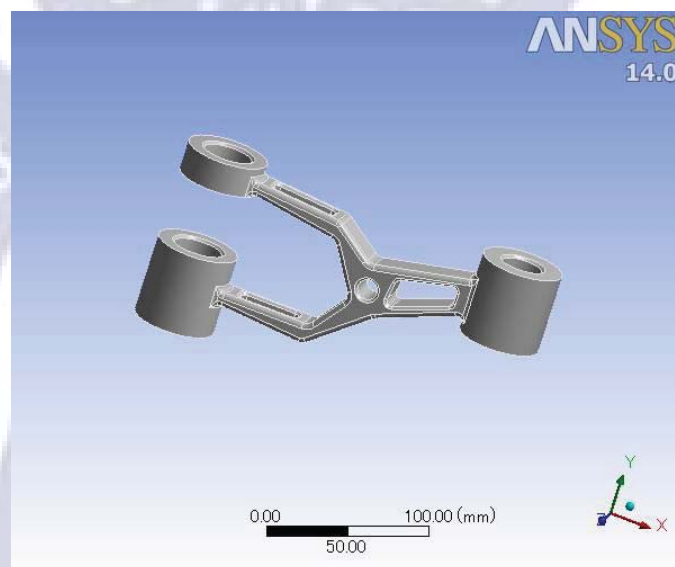


Figure B.1 Project

B.1 Units

Table B.2 Units

Unit System	Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius
Angle	Degrees
Rotational Velocity	rad/s
Temperature	Celsius

B.2 Model (A4)

B.2.1 Geometry

Table B.3 Model (A4) > Geometry

Object Name	<i>Geometry</i>
State	Fully Defined
Definition	
Source	C:\ANSYS\ycouple.stp
Type	Step
Length Unit	Meters
Element Control	Program Controlled
Display Style	Body Color
Bounding Box	
Length X	247.65 mm
Length Y	114.3 mm
Length Z	50.8 mm
Properties	
Volume	2.4318e+005 mm ³
Mass	1.9089 kg
Scale Factor Value	1.
Statistics	
Bodies	1
Active Bodies	1
Nodes	9731
Elements	5167
Mesh Metric	None

Table B.4 Model (A4) > Geometry (Continue)

Basic Geometry Options	
Solid Bodies	Yes
Surface Bodies	Yes
Line Bodies	No
Parameters	Yes
Parameter Key	DS
Attributes	No
Named Selections	No
Material Properties	No
Advanced Geometry Options	
Use Associativity	Yes
Coordinate Systems	No
Reader Mode Saves Updated File	No
Use Instances	Yes
Smart CAD Update	No
Attach File Via Temp File	Yes
Temporary Directory	C:\Users\000001A005DE4\AppData\Local\Temp
Analysis Type	3-D
Mixed Import Resolution	None
Decompose Disjoint Faces	Yes
Enclosure and Symmetry Processing	Yes

Table B.5 Model (A4) > Geometry > Parts

Object Name	<i>Part 1</i>
State	Meshed
Graphics Properties	
Visible	Yes
Transparency	1
Definition	
Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
Material	
Assignment	Structural Steel
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
Bounding Box	
Length X	247.65 mm
Length Y	114.3 mm
Length Z	50.8 mm
Properties	
Volume	2.4318e+005 mm ³
Mass	1.9089 kg
Centroid X	12.376 mm
Centroid Y	34.537 mm
Centroid Z	12.7 mm
Moment of Inertia Ip1	1697.7 kg·mm ²
Moment of Inertia Ip2	14544 kg·mm ²
Moment of Inertia Ip3	15632 kg·mm ²

Table B.6 Model (A4) > Geometry > Parts (Continue)

Statistics	
Nodes	9731
Elements	5167
Mesh Metric	None

B.2.2 Coordinate Systems**Table B.7** Model (A4) > Coordinate Systems > Coordinate System

Object Name	<i>Global Coordinate System</i>
State	Fully Defined
Definition	
Type	Cartesian
Coordinate System ID	0.
Origin	
Origin X	0. mm
Origin Y	0. mm
Origin Z	0. mm
Directional Vectors	
X Axis Data	[1. 0. 0.]
Y Axis Data	[0. 1. 0.]
Z Axis Data	[0. 0. 1.]

B.2.3 Mesh

Table B.8 Model (A4) > Mesh

Object Name	<i>Mesh</i>
State	Solved
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	0.697820 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre
View Advanced Options	No
Patch Conforming Options	
Triangle Surface Mesher	Program Controlled

Table B.9 Model (A4) > Mesh (Continue)

Advanced	
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Statistics	
Nodes	9731
Elements	5167
Mesh Metric	None

B.2.4 Static Structural (A5)**Table B.10** Model (A4) > Analysis

Object Name	<i>Static Structural (A5)</i>
State	Solved
Definition	
Physics Type	Structural
Analysis Type	Static Structural
Solver Target	Mechanical APDL

Table B.11 Model (A4) > Analysis (Continue)

Options	
Environment Temperature	22. °C
Generate Input Only	No

Table B.12 Model (A4) > Static Structural (A5) > Analysis Settings

Object Name	<i>Analysis Settings</i>
State	Fully Defined
Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Program Controlled
Large Deflection	Off
Inertia Relief	Off
Restart Controls	
Generate Restart Points	Program Controlled
Retain Files After Full Solve	No

Table B.13 Model (A4) > Static Structural (A5) > Analysis Settings (Continue)

Nonlinear Controls	
Force Convergence	Program Controlled
Moment Convergence	Program Controlled
Displacement Convergence	Program Controlled
Rotation Convergence	Program Controlled
Line Search	Program Controlled
Stabilization	Off
Output Controls	
Stress	Yes
Strain	Yes
Nodal Forces	No
Contact Miscellaneous	No
General Miscellaneous	No
Calculate Results At	All Time Points
Max Number of Result Sets	Program Controlled

Table B.14 Model (A4) > Static Structural (A5) > Analysis Settings (Continue)

Analysis Data Management	
Solver Files Directory	C:\Users\000001A005DE4\AppData\Local\Temp\WB_MN-MCAD09_2332_2\unsaved_project_files\dp0\SYS\MECH\
Future Analysis	None
Scratch Solver Files Directory	
Save MAPDL db	No

Table B.15 Model (A4) > Static Structural (A5) > Analysis Settings (Continue)

Analysis Data Management	
Delete Unneeded Files	Yes
Nonlinear Solution	No
Solver Units	Active System
Solver Unit System	mm

Table B.16 Model (A4) > Static Structural (A5) > Loads

Object Name	<i>Force</i>	<i>Fixed Support</i>
State	Fully Defined	
Scope		
Scoping Method	Geometry Selection	
Geometry	1 Face	2 Faces
Definition		
Type	Force	Fixed Support
Define By	Components	
Coordinate System	Global Coordinate System	
X Component	1000. N (ramped)	
Y Component	0. N (ramped)	
Z Component	0. N (ramped)	
Suppressed	No	

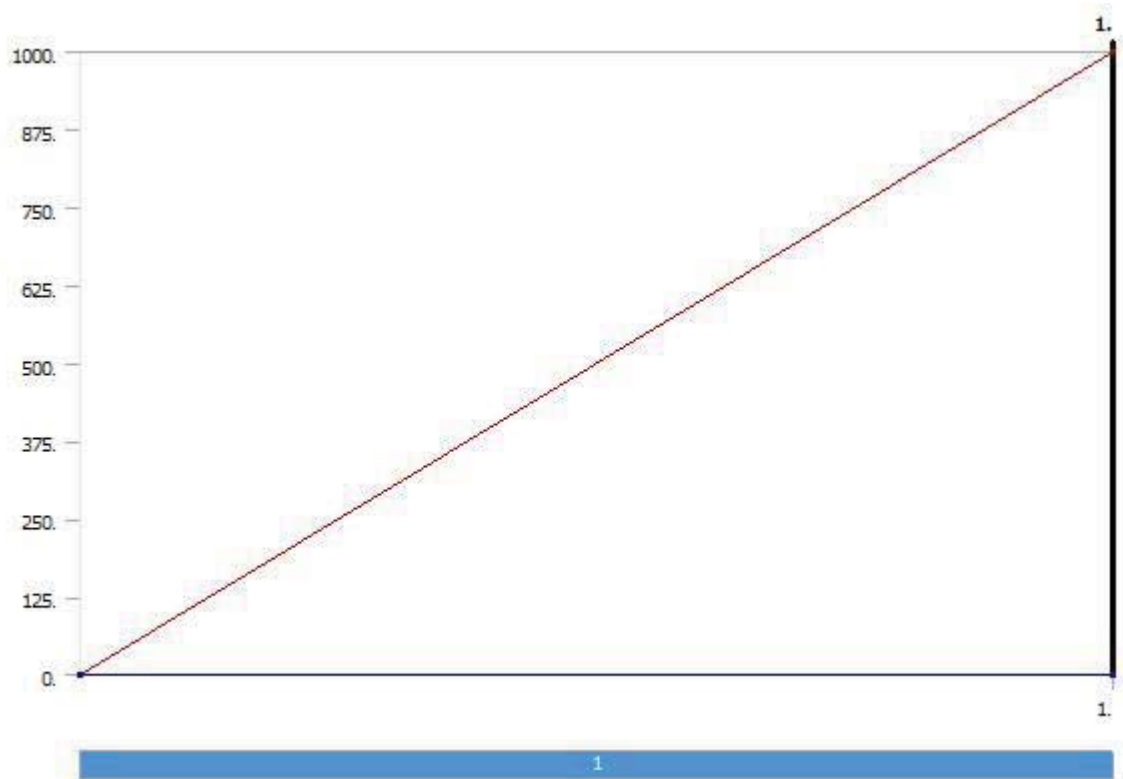


Figure B.2 Model (A4) > Static Structural (A5) > Force

Solution (A6)

Table B.17 Model (A4) > Static Structural (A5) > Solution

Object Name	<i>Solution (A6)</i>
State	Solved
Adaptive Mesh Refinement	
Max Refinement Loops	1.
Refinement Depth	2.
Information	
Status	Done

Table B.18 Model (A4) > Static Structural (A5) > Solution (A6) > Solution Information

Object Name	<i>Solution Information</i>
State	Solved
Solution Information	
Solution Output	Solver Output
Newton-Raphson Residuals	0
Update Interval	2.5 s
Display Points	All
FE Connection Visibility	
Activate Visibility	Yes
Display	All FE Connectors
Draw Connections Attached To	All Nodes
Line Color	Connection Type
Visible on Results	No
Line Thickness	Single
Display Type	Lines

Table B.19 Model (A4) > Static Structural (A5) > Solution (A6) > Results

Object Name	<i>Equivalent Stress</i>	<i>Vector Principal Stress</i>	<i>Total Deformation</i>
State	Solved		
Scope			
Scoping Method	Geometry Selection		
Geometry	All Bodies		

Table B.20 Model (A4) > Static Structural (A5) > Solution (A6) > Results (Continue)

Definition			
Type	Equivalent (von-Mises) Stress	Vector Principal Stress	Total Deformation
By	Time		
Display Time	Last		
Calculate Time History	Yes		
Identifier			
Suppressed	No		
Integration Point Results			
Display Option	Averaged		
Results			
Minimum	1.124e-004 MPa		0. mm
Maximum	17.772 MPa		2.0966e-002 mm
Information			
Time	1. s		
Load Step	1		
Substep	1		
Iteration Number	1		

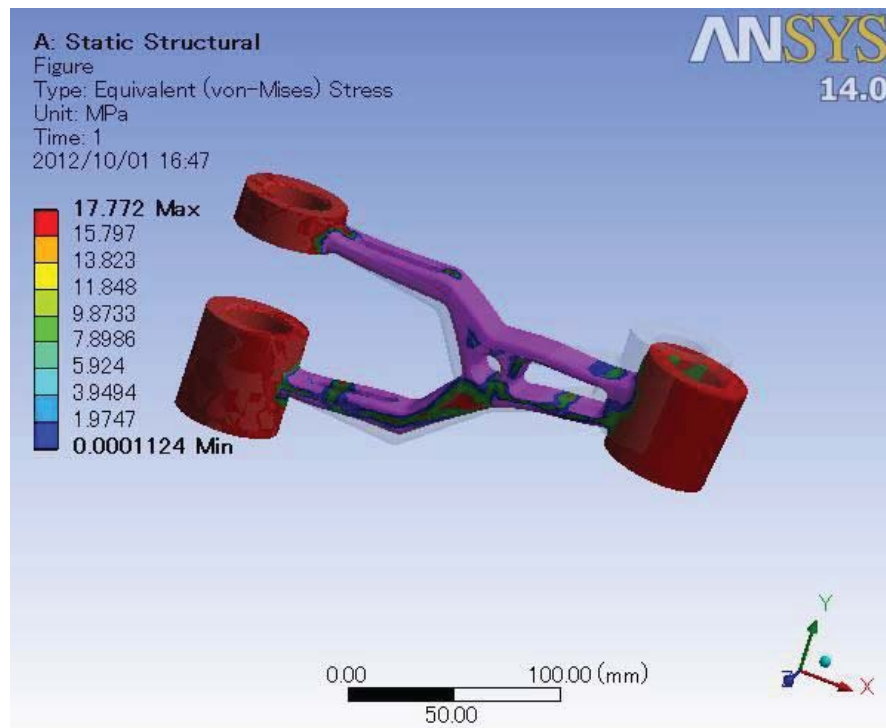


Figure B.3 Model (A4) > Static Structural (A5) > Solution (A6) > Equivalent Stress > Figure

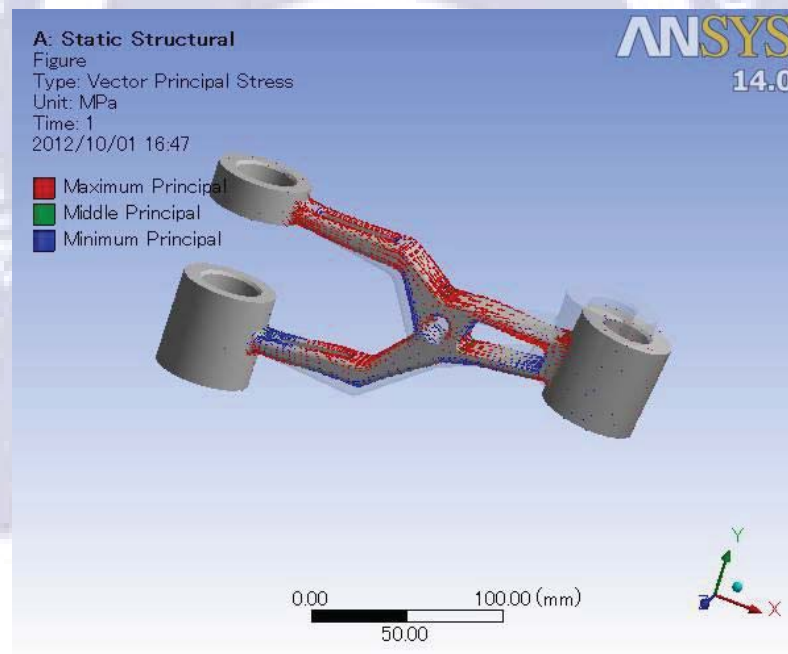


Figure B.4 Model (A4) > Static Structural (A5) > Solution (A6) > Vector Principal Stress >

Figure

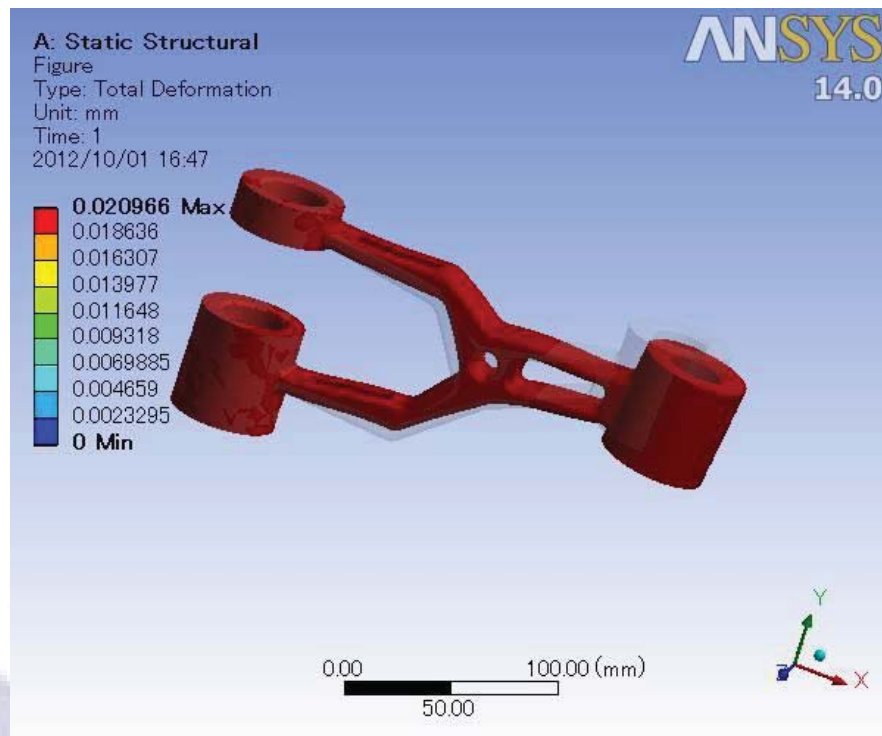


Figure B.5 Model (A4) > Static Structural (A5) > Solution (A6) > Total Deformation > Figure

B.3 Material Data

Structural Steel

Table B.21 Structural Steel > Constants

Density	7.85e-006 kg mm ⁻³
Coefficient of Thermal Expansion	1.2e-005 C ⁻¹
Specific Heat	4.34e+005 mJ kg ⁻¹ C ⁻¹
Thermal Conductivity	6.05e-002 W mm ⁻¹ C ⁻¹
Resistivity	1.7e-004 ohm mm

Table B.22 Structural Steel > Compressive Ultimate Strength

Compressive Ultimate Strength MPa
0

Table B.23 Structural Steel > Compressive Yield Strength

Compressive Yield Strength MPa
250

Table B.24 Structural Steel > Tensile Yield Strength

Tensile Yield Strength MPa
250

Table B.25 Structural Steel > Tensile Ultimate Strength

Tensile Ultimate Strength MPa
460

Table B.26 Structural Steel > Isotropic Secant Coefficient of Thermal Expansion

Reference Temperature C
22

Table B.27 Structural Steel > Isotropic Relative Permeability

Relative Permeability
10000

Table B.28 Structural Steel > Alternating Stress Mean Stress

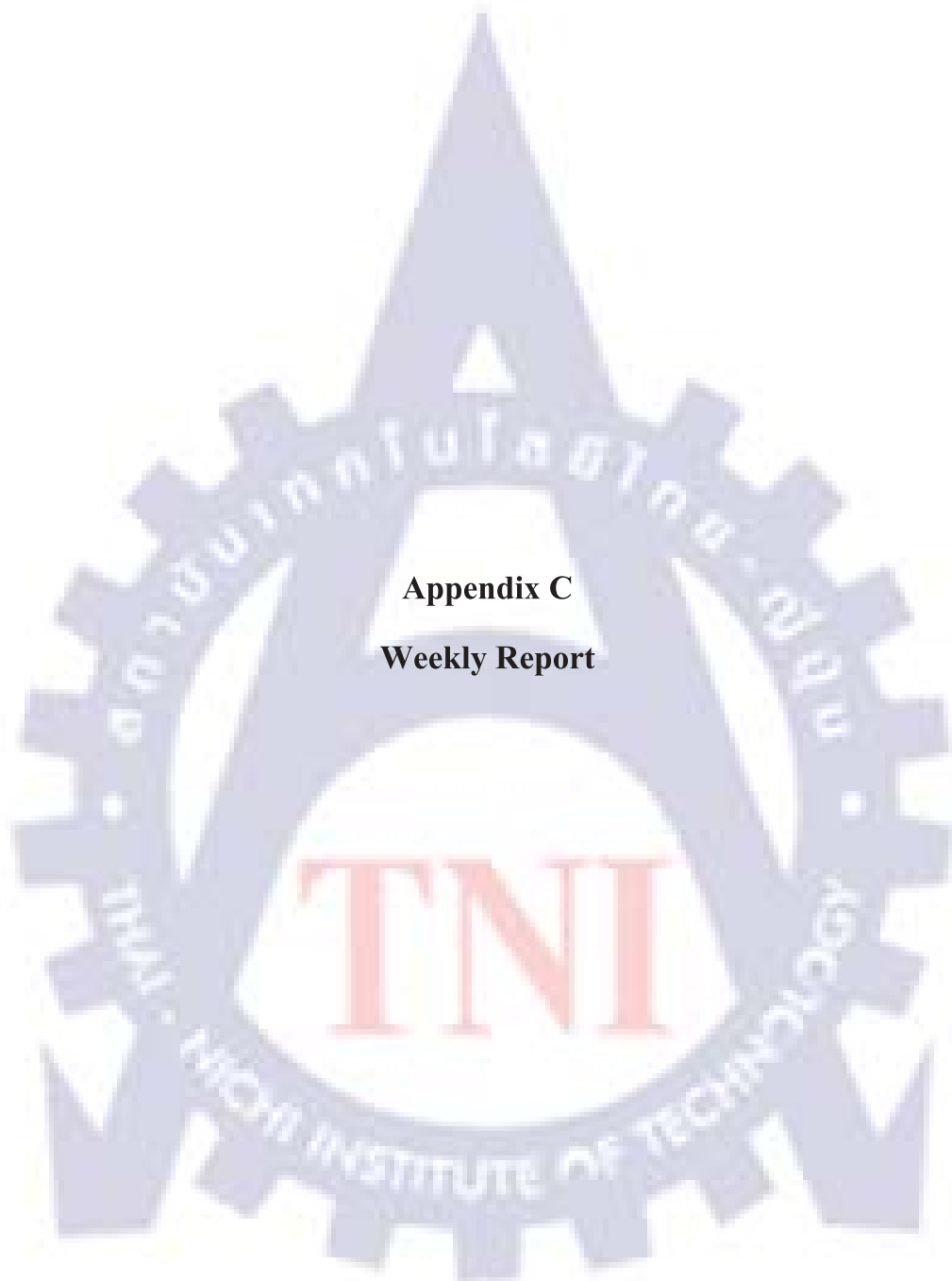
Alternating Stress MPa	Cycles	Mean Stress MPa
3999	10	0
2827	20	0
1896	50	0
1413	100	0
1069	200	0
441	2000	0
262	10000	0
214	20000	0
138	1.e+005	0
114	2.e+005	0
86.2	1.e+006	0

Table B.29 Structural Steel > Strain-Life Parameters

Strength Coefficient MPa	Strength Exponent	Ductility Coefficient	Ductility Exponent	Cyclic Strength Coefficient MPa	Cyclic Strain Hardening Exponent
920	-0.106	0.213	-0.47	1000	0.2

Table B.30 Structural Steel > Isotropic Elasticity

Temperature C	Young's Modulus MPa	Poisson's Ratio	Bulk Modulus MPa	Shear Modulus MPa
	2.e+005	0.3	1.6667e+005	76923



Appendix C
Weekly Report



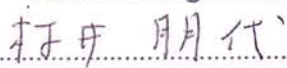
Co-operative Education and Career Center

1771/1, Pattanakarn Rd, Suan Luang, Bangkok, 10250, THAILAND Tel:0-2763-275 Fax : 0-2763-2754

Weekly Report Cooperative Education Work Plan

Student Name Nathani Smithimedhin Student ID 52113071-6
 Major Computer Engineering Faculty Engineering
 Company / Organization NEC Corporation (Tamagawa Plant)
 Advisor Name Murai Tomoyo Position Assistant Manager
 Tel +81 44 455-8391 E-mail t-murai@ap.jp.nec.com
 Cooperative education work at the organization 3D CAD modeling, Mechanical Analysis, Environment testing

Cooperative Education Work Plan (Week 1)

Date	Hours	Detail job description / general tasks	Knowledge	Problem
Monday 27/AUG/2012	(8.45~17.35) 8	Orientation Set up PC, email.		
Tuesday 28/AUG/2012	(8.45~17.35) 8	Make a comparison table of ODU (outdoor unit)		
Wednesday 29/AUG/2012	(8.45~17.35) 8	Learning about IP Code (International standard of protection)	What is IP Code? What does it mean?	
Thursday 30/AUG/2012	(8.45~17.35) 8	Make a comparison table of Environment test. Doing an Environment Test (IP test).	How to test an IPX6 standard.	
Friday 31/AUG/2012	(8.45~17.35) 8	Writing an IP test report. Reading a CREO software tutorial.	Basic using of CREO.	
Saturday/...../.....				
Sunday/...../.....				
Total Hours For this report	40	Student's signature  (MUR Nathani Smithimedhin) 31 / 08 / 2012	Advisor's signature  (Tomoyo Murai) Position <u>Assistant Manager</u> 31 / 8 / 2012	
Total Hours In last report	-			
Total Hours	40			



Co-operative Education and Career Center

1771/1, Pattanakarn Rd, Suan Luang, Bangkok, 10250, THAILAND Tel:0-2763-275 Fax : 0-2763-2754

Weekly Report Cooperative Education Work Plan

Student Name Nathari Smithimedhin Student ID 92113071-6
 Major Computer Engineering Faculty Engineering
 Company / Organization NEC Corporation (Tamagawa Plant)
 Advisor Name Mr. Tomoyo Murai Position Assistant Manager
 Tel +81-44-455-8391 E-mail t-murai@ap.jp.nec.com
 Cooperative education work at the organization 3D CAD Modeling, Mechanical Analysis, Environment testing

Cooperative Education Work Plan
(Week 2)

Date	Hours	Detail job description / general tasks	Knowledge	Problem
Monday 3./SEP./2012	8 (8.45-17.35)	Learning Basic of CREO and how to build a system by antenna, GMT, ODU.	Basic knowledge of CREO.	
Tuesday 4./SEP./2012	8 (8.45-17.35)	Drawing a Hexlink by CREO How to build a system by antenna 2 feet.		
Wednesday 5./SEP./2012	8 (8.45-17.35)	Learning about how to build a system by antenna 3 ft., How to move an antenna.		
Thursday 6./SEP./2012	8 (8.45-17.35)	Drawing 3D Bracket by using CREO.	More advance knowledge of CREO.	
Friday 7./SEP./2012	8 (8.45-17.35)	Drawing 3D Bracket by using CREO. ANSYS software tutorial.		
Saturday/...../.....				
Sunday/...../.....				
Total Hours For this report	40	Student's signature <u>ณัฐริ สมิตimedhin</u> (นายณัฐริ สมิตimedhin) <u>07 / 09 / 2012</u>	Advisor's signature <u>ฟู ฟู 朋代</u> (Tomoyo Murai) Position <u>Assistant Manager</u> <u>7 / 9 / 2012</u>	
Total Hours In last report	40			
Total Hours	80			



Co-operative Education and Career Center

1771/1, Pattanakarn Rd, Suan Luang, Bangkok, 10250, THAILAND Tel:0-2763-275 Fax : 0-2763-2754

Weekly Report Cooperative Education Work Plan

Student Name Nathari Smitthimedhin Student ID 521130711-6
 Major Computer Engineering Faculty Engineering
 Company / Organization NEC Corporation (Tamagawa Plant)
 Advisor Name Ms. Tomoyo Murai Position Assistant Manager
 Tel #1 44-455-8391 E-mail t-murai@ap.j.p.nec.com
 Cooperative education work at the organization 3D CAD Modeling, Mechanical Analysis, Environment test

Cooperative Education Work Plan

(Week 3)

Date	Hours	Detail job description / general tasks	Knowledge	Problem
Monday 10 / SEP / 2012	8 (8.45-17.30)	- Drawing 3D Door by using CREO. - Finish drawing bracket. - ANSYS tutorial.	How to use ANSYS to test a product.	
Tuesday 11 / SEP / 2012	8 (8.45-17.30)	- Assembly a door. - Learning how to build a mobile base device.		
Wednesday 12 / SEP / 2012	8 (8.45-17.30)	- Making a presentation of TIEROL Installation manual. -		
Thursday 13 / SEP / 2012	8 (8.45-17.30)	- Drawing 3D Baseband by using CREO. - Making a TIEROL installation manual. - Vibration		
Friday 14 / SEP / 2012	8 (8.45-17.30)	- Vibration Testing for Network device. - Drawing 3D Baseband by using CREO.		
Saturday / /				
Sunday / /				
Total Hours For this report	40	Student's signature <u>นัทธารีย์ สมิทธิเมธิน</u> (นางสาวนัทธารีย์ สมิทธิเมธิน) <u>14 / 09 / 2012</u>	Advisor's signature <u>Tomoyo Murai</u> (Tomoyo Murai) Position <u>Assistant Manager</u> <u>14 / 9 / 2012</u>	
Total Hours In last report	80			
Total Hours	120			



Co-operative Education and Career Center

1771/1, Pattanakarn Rd, Suan Luang, Bangkok, 10250, THAILAND Tel:0-2763-275 Fax : 0-2763-2754

Weekly Report Cooperative Education Work Plan

Student Name Natnari Smithimedhin Student ID 52113077-6
 Major Computer Engineer Faculty Engineering
 Company / Organization NEC Corporation (Tamagawa Plant)
 Advisor Name Mr. Tomoyo Murai Position Assistant Manager
 Tel +81-44-455-8391 E-mail t-murai@ap.jp.nec.com
 Cooperative education work at the organization 3D CAD Modeling, Mechanical Analysis, Environment test.

Cooperative Education Work Plan

(Week 4)

Date	Hours	Detail job description / general tasks	Knowledge	Problem
Monday 17.../SEP./2012	-	National Holiday	-	-
Tuesday 18.../SEP./2012	(8.45-17.00) 8	Drawing 3D Base band		
Wednesday 19.../SEP./2012	(8.45-17.00) 2	Drawing 3D Base band		
Thursday 20.../SEP./2012	(8.45-17.00) 6	Drawing 3D Base band		
Friday 21.../SEP./2012	-	NEC Holiday	-	-
Saturday/...../.....				
Sunday/...../.....				
Total Hours For this report	16	Student's signature <u>ณัฐณริย์ สัมภักดิ์เมธินกร</u> (นางสาว ณัฐณริย์ สัมภักดิ์เมธินกร) <u>20 / SEP / 2012</u>	Advisor's signature <u>トモヨ 村代</u> (Tomoyo Murai) Position <u>Assistant Manager</u> <u>20 / 9 / 2012</u>	
Total Hours In last report	120			
Total Hours	136			



Co-operative Education and Career Center

1771/1, Pattanakarn Rd, Suan Luang, Bangkok, 10250, THAILAND Tel:0-2763-275 Fax : 0-2763-2754

Weekly Report Cooperative Education Work Plan

Student Name Natnari Smithimedhin Student ID 52113071-6
 Major Computer Engineer Faculty Engineering
 Company / Organization NEC Corporation (Tamagawa plant)
 Advisor Name Mr. Tomoyo Murai Position Assistant Manager
 Tel +81-44-455-8391 E-mail t-murai@ap.jp.nec.com
 Cooperative education work at the organization 3D CAD Modeling, Mechanical Analysis Environment

Cooperative Education Work Plan
(Week 5)

Date	Hours	Detail job description / general tasks	Knowledge	Problem
Monday 24 / SEP / 2012	8 (8.45-17.30)	Making a report for vibration test. Assembly 1 Apply a 30 dBv band		
Tuesday 25 / SEP / 2012	8 (8.45-17.30)	Writing Report.		
Wednesday 26 / SEP / 2012	8 (8.45-17.30)	Writing Report.		
Thursday 27 / SEP / 2012	8 (8.45-17.30)	Writing Report.		
Friday 28 / SEP / 2012	8 (8.45-17.30)	Writing Report.		
Saturday / /				
Sunday / /				
Total Hours For this report	40	Student's signature <u>ณัฐนารี สมิทธิเมธิน</u> (นางสาวณัฐนารี สมิทธิเมธิน) <u>28 / SEP / 2012</u>	Advisor's signature <u>โทมโย มุไร</u> (Tomoyo Murai) Position <u>Assistant Manager</u> <u>28 / 9 / 2012</u>	
Total Hours In last report	136			
Total Hours	176			



Co-operative Education and Career Center

1771/1, Pattanakarn Rd, Suan Luang, Bangkok, 10250, THAILAND Tel:0-2763-275 Fax : 0-2763-2754

Weekly Report Cooperative Education Work Plan

Student Name Nathari Smithimedhin Student ID 52113091-6
 Major Computer Engineer Faculty Engineering
 Company / Organization NEC Corporation (Tama-gawa Plant)
 Advisor Name Mr. Tomoyo Murai Position Assistant Manager
 Tel +81-44-955-8391 E-mail t-murai@ap.jp.nec.com
 Cooperative education work at the organization 3D CAD Modeling, Mechanical Analysis, Environment test

Cooperative Education Work Plan
(Week 6)

Date	Hours	Detail job description / general tasks	Knowledge	Problem
Monday 1 / Oct / 2012	(8.45-17.35) 8	Writing Report		
Tuesday 2 / Oct / 2012	(8.45-17.35) 8	Writing Report		
Wednesday 3 / Oct / 2012	(8.45-17.35) 8	Writing Report Making a presentation		
Thursday 4 / Oct / 2012	(8.45-17.35) 8	Making a presentation		
Friday 5 / Oct / 2012	(8.45-17.35) 8	Presentation		
Saturday/...../.....				
Sunday/...../.....				
Total Hours For this report	40	Student's signature <u>ณัฐริ สมิตimedhin</u> (นางสาวณัฐริ สมิตimedhin) <u>5 / Oct / 2012</u>	Advisor's signature <u>村尾 朋代</u> (Tomoyo Murai) Position <u>Assistant Manager</u> <u>5.1.10.120/2</u>	
Total Hours In last report	176			
Total Hours	216			

Biography



Name-Surname Natnari Smitthimedhin

Date of Birth 02 June 1991

Education Background

Primary education Wattana Wittaya Academy
May 1996

Secondary education Wattana Wittaya Academy
May 2003

Undergraduate education Thai-Nichi Institute of Technology
Computer Engineering June 2009

Seminar Background:

1. Japanese Language and Culture at Aomori Chuo Gakuin University, Aomori, Japan
2. Japanese Language and Electrical in Robot Laboratory at Tohoku Institute of technology, Sendai, Japan