International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 02, February-2017

Modelling and Analysis of Drilling Jig for Mounting Casing of Electric Motor

Geet Prasanna Pandit,
Department of Mechanical Engineering,
DKTE'S Textile and Engineering Institute,
Ichalkaranji.

Sagar Bhausaheb Patil,
Department of Mechanical Engineering,
DKTE'S Textile and Engineering Institute,
Ichalkaranji.

Rushikesh Vashantrao Shinde,
Department of Mechanical Engineering,
DKTE'S Textile and Engineering Institute,
Ichalkaranji.

Shubham Baburao Amale
Department of Mechanical Engineering,
DKTE'S Textile and Engineering Institute,
Ichalkaranji.

***Abstract*— In today's fast growing industrial sector jig plays very important role by time saving to achieve quality work pieces. Use of drilling jig increases the production rate with good quality and accuracy. The main objective of using jigs and fixtures in an industry is to achieve the Interchangeable Part Concept, and these are mainly used where production of goods is on large scale. The scope of this paper is to study the performance parameter by modelling an assembly of Drill Jig in advanced modelling software like CATIAV5R20 and analysis was done by using the predominant analysis software ANSYS 14.5. For the study, this Drill Jig is designed for the 'Mounting casing of Electric Motor'. It involves the modelling of individual part, assembly of the drill jig and analysis is carried out on clamping system to determine the stress and deformation.**

Keywords*— Drill jig, Design, Modelling, Analysis, CATIA V5R20, ANSYS R14.5.

1. INTRODUCTION

Increasing the productivity and accuracy are the two basic aims of mass production. As we know the solution to this is by reducing the set up cost of the machine and also reducing the manual fatigue. New machine tools, high performance cutting tools, and modern manufacturing processes enable today's industries to make parts faster and better than ever before. Jigs and fixtures form an important category of equipment that goes a long way in achieving productivity. The purpose of jigs and fixtures is to maintain low manufacturing costs and to increase industrial efficiency. Its purpose is also to speed up machining times by eliminating time of handling and setting of the component parts. Further jigs and fixtures are making it possible to employ unskilled or semi- skilled operators in industry [1].

Jig is mainly used to guide the cutting tool in specific operations like drilling, rimming, etc. Also it holds, supports and locate the work piece. Jigs are, usually fitted with hardened steel bushings for guiding drills or other cutting tools. Holes are bored in the structure, so that when tools are fed through them and into the component, holes are made in the component in the correct positions as required by the component drawing [4]. It is usually necessary for the work to be held in the jig by clamping. Jig is usually not fixed to the machine table by clamping.

2. LITURATURE SURVEY

K. Rama Subba Reddy, S. Ramesh Kumar Babu and A.V. Hari Babu, "Design and Analysis of Drill Jig at Variable Materials" in the context of International Research Journal of Research and Technology (IRJRT) [2] has completed have Modelling and analysis by using advanced modelling software system CATIA and ANSYS respectively. Static structural and MODAL analysis was done on DRILL JIG for 3 completely different materials and results are achieved. They found that in line with the results stress is nearly shut for 3 materials however considering deformation is a smaller amount for steel compared with grey forged iron and aluminum alloy. Thus the steel is best material for the producing.

Abdullah Jasim Mohammed, Dr. Mohammed Tariq, "Design and Analysis of Drill Jig for a Shaft using ANSYS" [1]. In order to compete with the ever-growing competent market, it is indispensable for an industrialist to aim for higher production coupled with the enhanced quality with an objective of reduction in the cost of potential. In this paper they found that the usage of Drill jig for mass production is very easy and cost effective compared to any other technique. They have designed a jig for drilling purpose in a software package called PRO E and the drilling force is applied on the shaft and is calculated for the different loading conditions and compared with two different materials i.e. steel and aluminum with the software ansys. They concluded that with the help of the software tools such as PRO E and ANSYS one will be able to design any model accurately and tell exact stresses with much accuracy and in less time compared to manual calculation.

Shivanand Vathare, Shrinivas L Gombi, Darshan M Katgeri, "Design and Analysis of Drill Jig for Head and Cover Part of the Actuator" [3]. The objective of this project work is to design a drill jig for head and cover part of the cylinder actuator. The SOLIDEDGE V19 software is used to model the drill jig, and analysis work is carried out on clamp plates to determine the stress, strain and deformation by using SOLIDWORKS and ANSYS Workbench. After the analysis they found that the stress, strain and deformation of clamp plates were well below the allowable limits.

As per the companies present requirement they need such a technique for drilling operation which can be efficiently used to reduce the cost of production, improve the quality of the product, increase the production rate and reduce the operation time.

In this paper we have modeled the Drill Jig for 'Mounting Casing of Electric Motor' for drilling 3 holes of 4 mm diameter. These 3 holes won't be drilled simultaneously. But with special purpose machine we can drill 3 holes simultaneously to increase productivity.

As per the dimensions we modeled the part on CATIA V5R20.

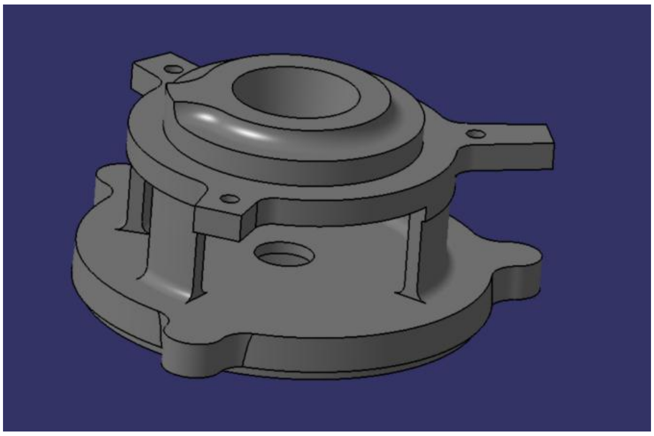

Fig. 1 CAD Model of Mounting Casing

3. METHODOLOGY

The main objective of the study is to check whether the Drill Jig is withstanding the load applied during clamping the work piece or not.

So the methodology of the study includes

1. CAD Model of Mounting casing using CATIAV5
2. Design of Drill Jig
3. CAD Model of Drill Jig assembly
4. Meshing of Drill Jig using ANSYS R14.5
5. Elemental analysis at various load.
6. Result and Conclusion

4. MODELLING OF DRILL JIG ASSEMBLY

3D Modelling is used in a variety of applications to make representations of physical objects on the computer. 3D modelling is a subset of Computer Aided Design (CAD), in which you use a computer to assist in the design process for any type of design work. It is used in a variety of applications, mostly when it comes to designing parts on the computer to assist in the making or visualization of those parts. The computer model is used to communicate dimensions, material types, etc. to anyone viewing the design.

CATIA (computer aided three-dimensional interactive application) is a multi-platform software suite developed by the French company Dassault Systems. CATIA enables the creation of 3D parts, from 2D sketches, sheet metal, composites, and moulded, forged or tooling parts up to the definition of mechanical assemblies. The software provides advanced technologies for mechanical surfacing. It provides tools to complete product definition, including functional tolerances as well as kinematics definition. CATIA provides a wide range of applications for tooling design, for both generic tooling and mould & die.

Post Clamp is used for clamping the product which consists of C- washer, Nut, Bolt etc. Each part of Drill Jig is modelled separately on CATIAV5 like Jig plate, Locator, C-washer, Nut, Bolt etc. These parts are then assembled in Assembly workbench by applying different constraints.

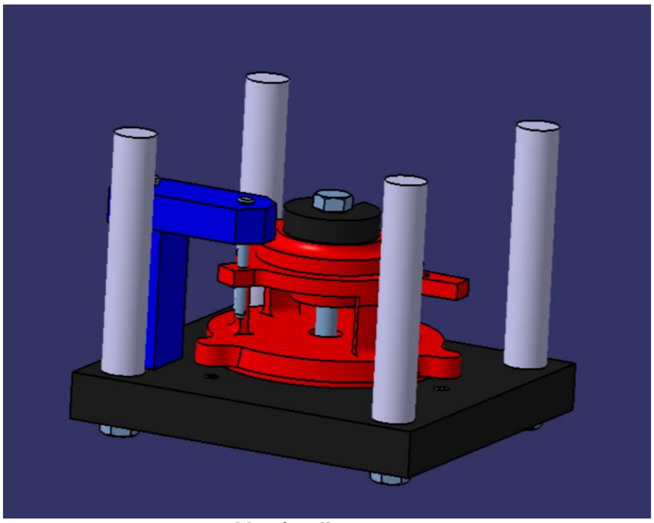

Fig. 2 Assembly of Drill Jig in CATIA V5R20

5. ANALYSIS OF DRILL JIG ASSEMBLY

The finite element method (FEM) is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. It is also referred to as finite element analysis (FEA). It subdivides a large problem into smaller, simpler parts that are called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem. FEM then uses variation methods from the calculus of variations to approximate a solution by minimizing an associated error function [2].

The ANSYS program is self-contained general purpose finite component program developed and maintained by Gloria May Josephine Svensson Analysis Systems Iraqi National Congress. The program contains several routines, all reticulated and every one for main purpose of achieving a solution to an engineering drawback by Finite component methodology. ANSYS provides an entire resolution to design issues. It consists of powerful design capabilities like full constant quantity solid modelling, design optimization and automotive vehicle meshing, which provides engineers full management over their analysis [2].

Analysis of Drilling Jig

In this paper the Drilling Jig is analyzed against the clamping force which is applied while clamping the work piece (Mounting Casing).

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 02, February-2017

The clamping force is developed by the screw clamp can be calculated with the following formula_

$$Fs = \frac{Fh \times L}{R\tan(\propto + \emptyset)}$$

Where,
Fs = Force developed by screw
Fh = Pull or push applied to spanner
R = Pitch radius of screw thread
α= Helix angle of thread
ϕ= Friction angle of thread
L = Length of spanner [4]

By the calculation, we found that maximum force applied is 5000N. By considering Factor of Safety we analyzed the component at various load conditions i.e. at 5000N, 6000N and 7000N.

Static Structural Analysis

A static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads.

The figure shows the deformation and von mises stress diagram at various loads which were applied to the Drill Jig assembly. The graph shows the Force Convergence and force criterion plot at various load. The Convergence criteria defines how close to this exact balance is acceptable.

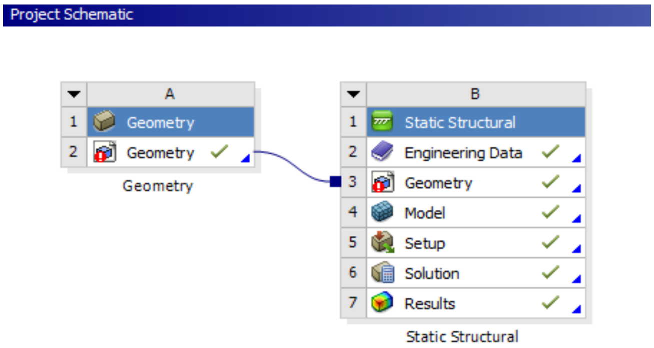

Fig.3 Project Schematic

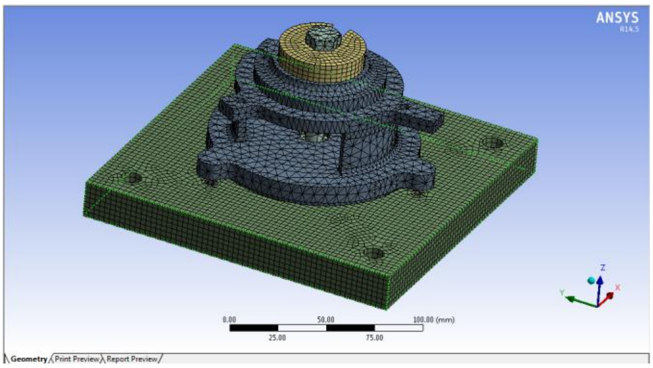

Fig.4 Meshed Assembly

Structural Analysis at 7000N load

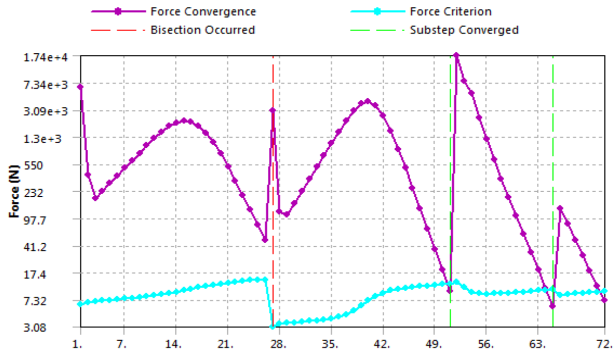

Fig. 5 Solution Information – Force Convergence

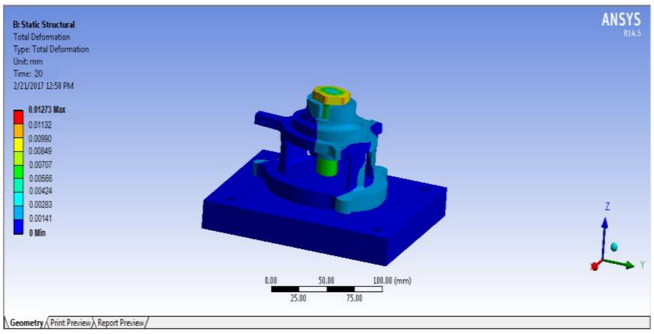

Fig.6 Deformation at 7000N load

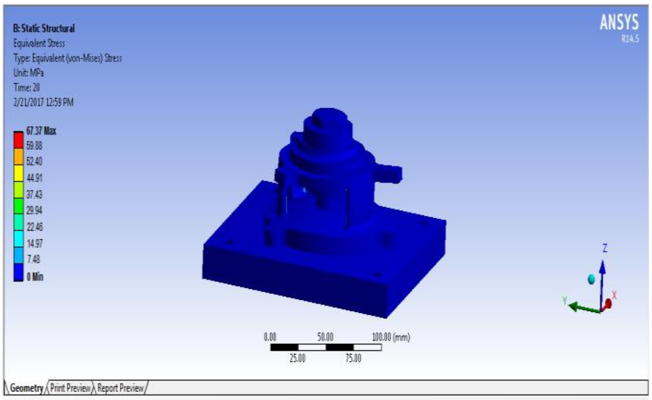

Fig.7 Equivalent (von-mises) stress at 7000N load

Structural Analysis at 6000N load

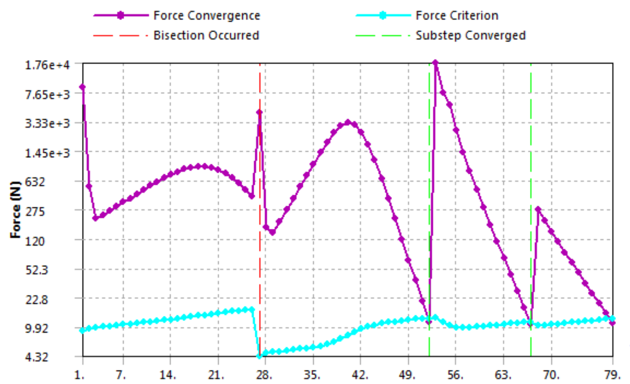

Fig.8 Solution Information – Force Convergence

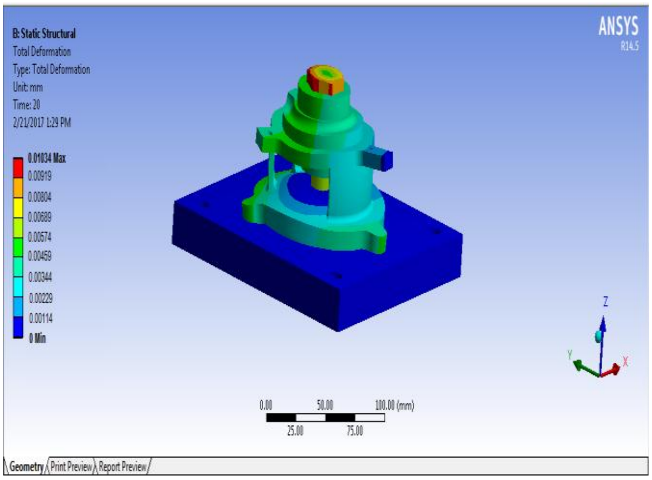

Fig.9 Deformation at 6000N load

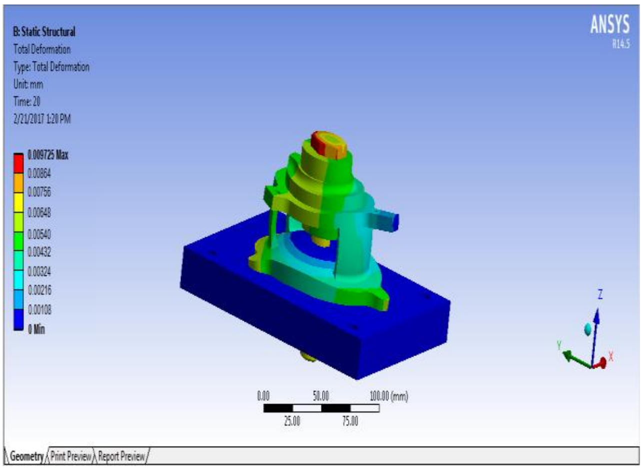

Fig.12 Deformation at 5000N load

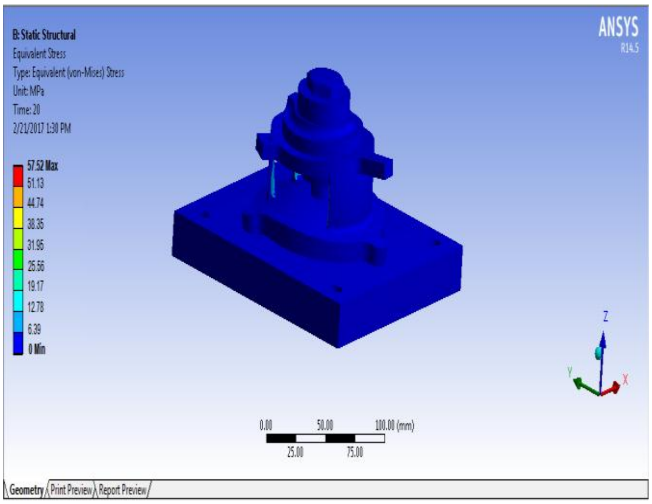

Fig.10 Equivalent (von-mises) stress at 6000N load

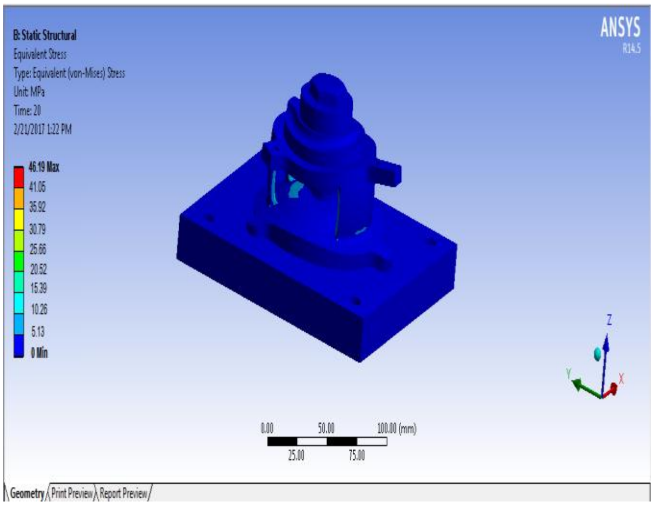

Fig.13 Equivalent (von-mises) stress at 5000N load

Structural Analysis at 5000N load

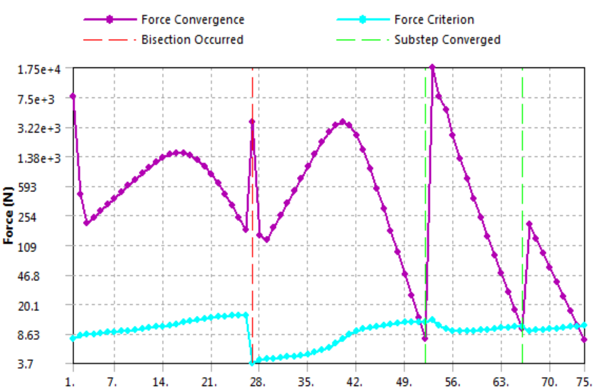

Fig.11 Solution Information – Force Convergence

6. RESULT

Table No.1

Load (N)	Max. Deformation(mm)	Von-Mises Stress (MPa)	Remark
7000	0.0127	67.37	SAFE
6000	0.0103	57.52	SAFE
5000	0.0972	46.19	SAFE

The maximum permissible value of stress of structural steel is 250 MPa.

From this it is clear that the design of Jig is safe up to 7000N as there is minimum acceptable deflection. The stress is also less than the permissible stress of the material. Hence design of Drill Jig Assembly is safe.

7. CONCLUSION

Drilling jig is used for mass production and effective cost operation. Drill jig is successfully designed for mounting casing of electric motor. The modelling of drill jig is by using CATIA V5R20 software. The drill jig provides interchangeability to the mounting casing. The critical part of the drill jig is the clamping system. For this analysis is done on post clamp by using ANSYS 14.5 software.

Based on the analysis results, following conclusion points are summarized,

- The maximum permissible value of stress of structural steel is 250 MPa.
- From the results achieved at loads 5000N, 6000N and 7000N it has given lower stress values and deformation for the structural steel.

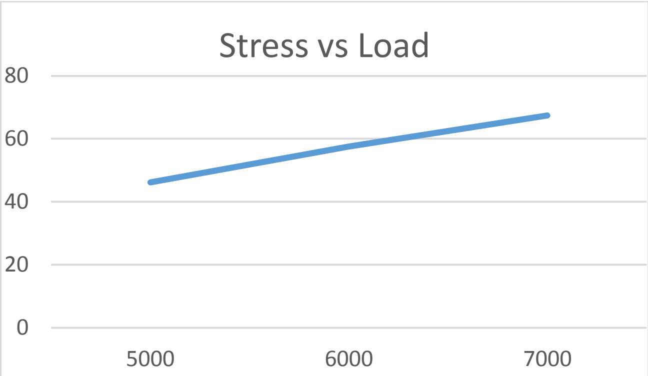

Fig. 14 Stress vs. Load Plot

- From this it can be stated that at peak load i.e. 7000N which is more than the clamping force the jig will with stand.

8. REFERENCES

[1] Abdullah Jasim Mohammed, Dr. Mohammed Tariq, "Design and Analysis of Drill Jig for a Shaft using ANSYS", International Journal of Scientific Engineering and Technology Research, Vol.02, Issue.18, December-2013, Pages: 2059-2066.

[2] K. Rama Subba Reddy, S. Ramesh Kumar Babu and A.V. Hari Babu, "Design and Analysis of Drill Jig at Variable Materials", International Research Journal of Engineering and Technology (IRJET), Volume: 03 Issue: 06, June-2016, Page 2022-2029.

[3] Shivanand Vathare, Shrinivas L Gombi, Darshan M Katgeri, "Design and Analysis of Drill Jig for Head and Cover Part of the Actuator", International Journal of Engineering Research & Technology (IJERT) Vol. 3 Issue 8, August – 2014, Pages-184-191.

[4] Nirav Newar, K. S. Chaitnaya, Phani Shankar, "Modelling and Analysis of Drill Jig", International Journal of Innovative Technologies (IJITECH) Vol. 3 Issue 10, November-2015, Pages- 1838-1844

[5] NBV Lakshmi Kumari, G. Prsasnna Kumar, "Design and Analysis of Indexing Type of Drill Jig", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 12, Issue 2 Ver. I (Mar - Apr. 2015), PP 46-51

[6] P H Joshi, "Jigs and fixtures" Tata McGraw-Hill Education, 2010

[7] Kempster , M.H.A., Principles of Jig and Tool Design, English Universities Press, 1968

[8] Kempster, M.H.A., Introduction to Jig and Fixture Design, 3rd edition, Hodder & Stroughton Ltd. 1978

[9] Colvin, F. H. and L. L. Hass, Jigs and Fixtures, Tata McGraw-Hill Publ. Co. Ltd., 1978