# IPTN IN-HOUSE UTILITY SOFTWARE FOR STRESS ANALYSIS PROCESS

M. Mochajan *)
Trias Santoso *)
John Herry Karo Karo *)

Presented at 1991 MSC World Users Conference
March 11-15, 1991

**Abstract**

Stress analysis process using FEM, beginning from input data preparation, model checking, validation to final result and documentation, should be perform effectively, efficient, and with good accuracy.

This paper describes the experiences in making utility software to support MSC/NASTRAN to achieve the above goals.

Using available software and hardware, the utility software is made mainly for stress analysis requirement. The significance of this software is it's user friendliness. It is graphic display oriented and integrated on one network system, so that all stress analysis procedures can be performed at one terminal.

*) Stress Analysis Department, Directorate of Technology
PT. IPTN ,Nusantara Aircraft Industry, INDONESIA

## 1. INTRODUCTION

Since MSC/NASTRAN became available at IPTN, large and complex structures have been analyzed quickly. In spite of that, experiences show that data preparation and output processing take much more time than the MSC/NASTRAN analysis itself.

IPTN developed in-house utility software, those are pre processor and post processor, to support MSC/NASTRAN so that efficiency and accuracy of the analysis can be increased.

Using available software and hardware, the utility software is made mainly for stress analysis requirement. The program is written in Fortran, Dialogue Management and Graphic Data Display Management Language. The main feature of this software is its friendliness. It is graphic display oriented and integrated on one network system, so that the complete stress analysis process can be performed at one terminal.

## 2. SOFTWARE DEVELOPMENT

At IPTN MSC/NASTRAN Version 66 is used mainly for stress analysis. The software is installed on IBM 3090-600S with the MVS/XA operating system. The general analysis procedure is described in figure 1.

1. **Input Data**
   Geometry and structure dimension data are recieved from the Structure Design Department. Load Data comes from the Aircraft Load Department, or for certain cases the loads are calculated by the Stress Analysis Department.

2. **Model Creation**
   Based on those data the model is generated by using CADAM/MESH. After that, the model is transferred to TSO for adding the Executive Control Deck, Case Control Deck and Bulk Data Deck. For a simple model, all work can be done directly on TSO.

3. **Model Checking**
The model is checked for completeness and correctness of all input data so that it is ready for execution. The main task in this step is to check the geometry and topology of the model by using graphic software.

4. **MSC/NASTRAN Execution**
MSC/NASTRAN is executed to find out the structure response due to the applied load.

5. **Validation**
Validation consists of checking matrix singularity, epsilon, displacement, forces on single point constraint and the load vector. For certain cases, element forces and stress pattern can also be used for validation. If the model has an error, it is corrected and executed again until valid results are obtained.

5. **Document preparation**
MSC/NASTRAN output, as well as other important information, are arranged in typical format for a document. In order to make easier for inspection, the numerical type of output is displayed graphically.

To perform these procedure easily, software has been developed by the staff of the Stress Analysis Department. The program was created especially to fulfill internal needs but it can also be used by another company. Currently the programs are developed using the available software and hardware in IPTN's IBM computer environment. All programs are integrated in one network, so that communication among the programs are easy and the software is user friendly.

## 3. INTEGRATED IN-HOUSE UTILITY SOFTWARE

The software developed by the Stress Analysis Department is integrated into one network that we call NUSAP (NUsantara Structural Analysis integrated Program). Nusap is divided into four groups, namely :

1. Finite Element Method
2. Stress Analysis Application Programs

3. Damage Tolerance Analysis Programs
4. Utility.

This paper only presents groups 1 and 2, ie Finite Element Method and Stress Analysis Aplication Programs.

The In-house utility programs which support stress analysis by MSC/NASTRAN are called DINAFEM, MOMENAX, MAMI, and NEO. Those software are used in the process of model checking, validation and document preparation. Figure 2 shows the overall diagram.

### 3.1 Model Checking

For model checking , DINAFEM Preprocessor reads MSC/NASTRAN input data and then displays them graphically. The user can look over the model easily using this software. Four categories are diagnosed, these are : geometry, load, aspect ratio and warping ratio.

For geometry checking, element connectivity is first inspected. DINAFEM has the facility to show undefined grid points and wrong connectivity. Another facility is called Display Edge which displays the edges of the model and the intersection line of two planes. By using it, missing plane element can be seen directly.

Element groupings can be created to perform geometry checking more easily. The user can display elements based on coordinate, element type, element ID and unique digit number. The user can also display element shrink and define colors for each type of elements. With these capabilities geometry inspection can be done efficiently. Figure 3 show the DINAFEM capability to perform grouping. Using four split displays the model can be describe in clear and complete view.

Load inspection can be performed by display location and direction of the applied load, this allows the user to determine whether it agrees with the specified load condition. The aspect ratio and warping ratio of plane elements can also be displayed, this facilitates the checking of element geometrical requirements.

## 3.2. Validation by Post Processor

Validation is done by graphically displaying MSC/NASTRAN output. The DINAFEM Post Processor utility performs this function. DINAFEM needs MSC/NASTRAN input and analysis output in the OUTPUT2 format. For this purpose an alter is included which writes displacement and stress data blocks to a file in OUTPUT2 format.

The capabilities to display deformed geometry and stress patterns can be utilized to validate Finite Element Model. Figure 4 is the deformed geometry of typical fuselage due to an emergency forward landing. Figure 5 is the stress pattern on the skin for the same model. Another software that can be employed for model validation is MOMENAX. This software reads MSC/NASTRAN Output and displays bending moment, axial force and shear force diagrams. Every displayed graph can be printed using the terminal screen hardcopy.

## 3.3. Document Preparation

To facilitate Document preparation, a software was developed to extract MSC/NASTRAN output, generate the document header and footer and reshape output format.

In the documentation general results of the analysis need to be described, for this purpose MAMI and NEO can be used. MAMI extracts maximum and minimum force values for every element type, every subcase and every force component from MSC/NASTRAN output. NEO identifies elements with stresses higher than a specified value.

MSC/NASTRAN output is a 133 character record which can not directly be printed on a A4 size paper. The data is transferred to a personal computer where character size is adjusted to fit the document format.

Utilizing the mentioned software, available hardware can be used effectively to produce documentation in a shorter time. In the coming years the software capability will be improved according to user's specifications and needs.

## 4. HARDWARE CONFIGURATION

Hardware configuration with related to the In-house Utility Software for Stress Analysis process shown in figure 6. Hardware avaliable at IPTN to operate the developed software are:

- IBM 3090-600S
- Control Unit IBM 3174
- Mux IBM 3299/2
- Terminal IBM 3179 G
- Printer Hardcopy IBM 3287/2c
- Hard Disk
- Emulator Card IBM PC 3270
- Computer IBM PS2 50Z
- Printer EPSON LX800.

## 5. CONCLUSION

This utility software proves to be cheap, powerful, and user friendly, and can be utilized to improve the efficiency of the stress analysis process.

The programmers are staff members of the Stress Analysis Departement.

## REFERENCES

1. GDDM Application Programming Guide, first edition, IBM, 1986.
2. Interactive System Productivity Facility (ISPF), Dialog Management Services, third edition, IBM, 1985
3. MSC/NASTRAN Programer's Manual, Volume 1, The MacNeal Schwendler Corporation, Los Angeles, CA, November 1980
4. MSC/NASTRAN User's Manual, Version 66, The MacNeal Schwendler Corporation, Los Angeles, CA, November 1988.
5. Santoso, T., DINAFEM User's Manual, IPTN, Bandung, 1990.
6. VS FORTRAN VERSION 2 PROGRAMMING GUIDE, Release 3, IBM Corporation, San Jose, CA, 1988

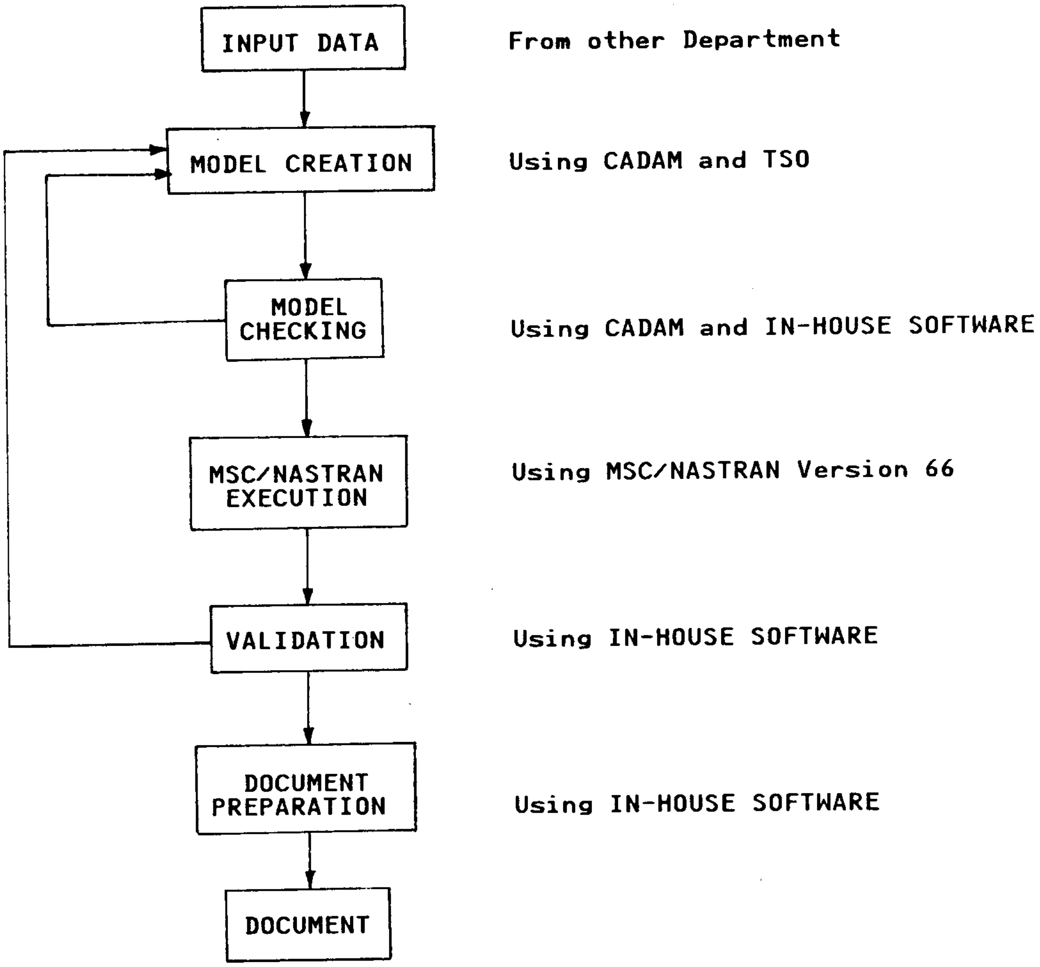


*Figure 1. Stress Analysis Procedure.*

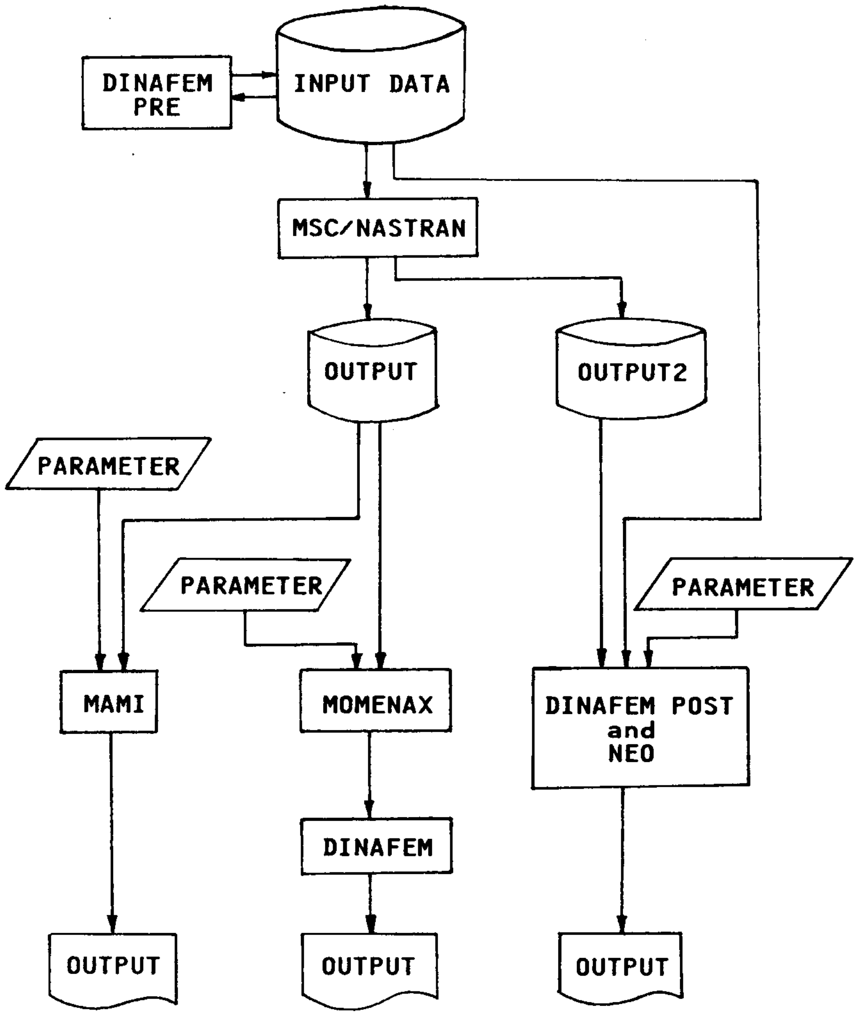


*Figure 2. General Flow of Analysis Process.*

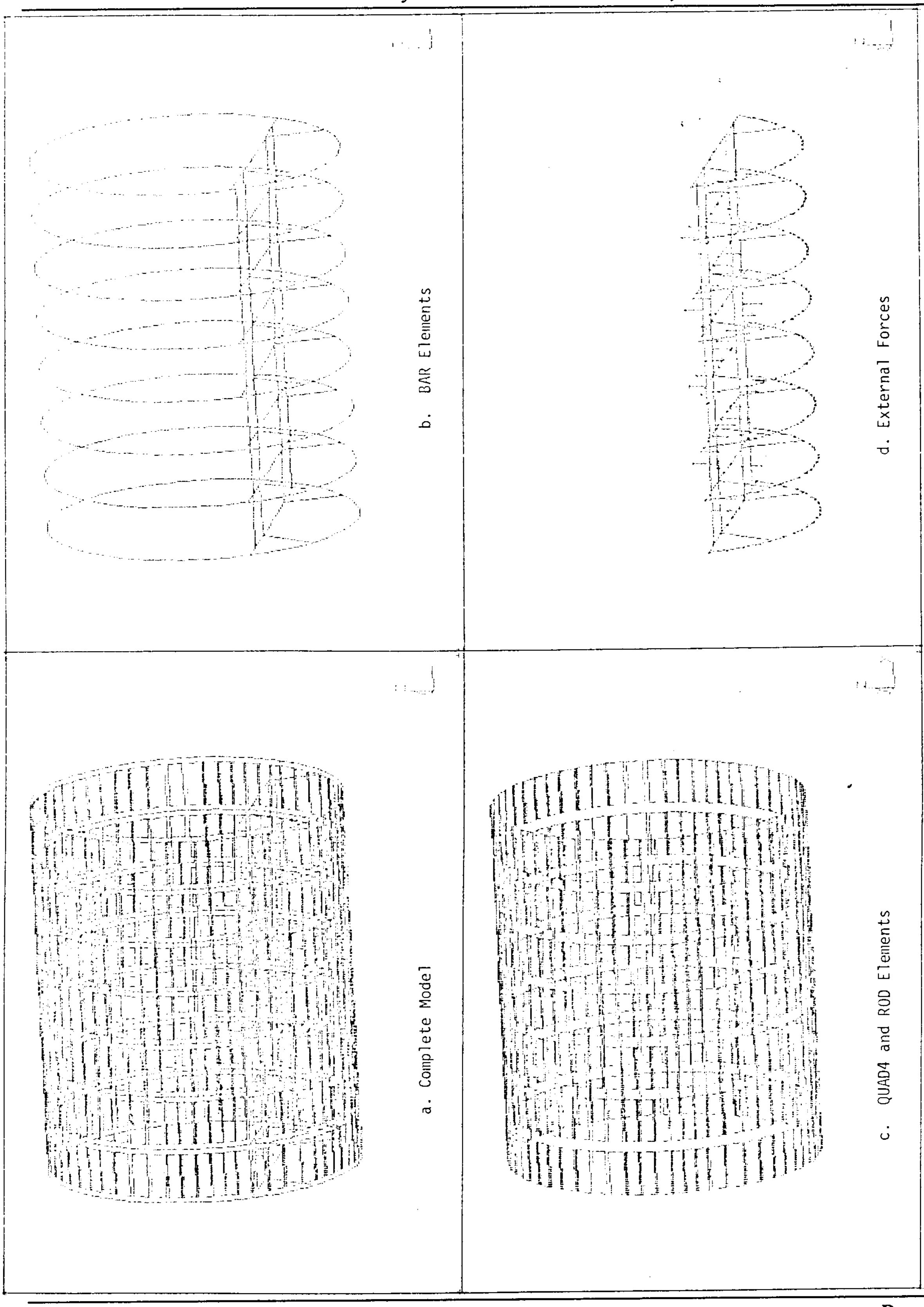


*Figure 3. Element Grouping by DINAFEM.*

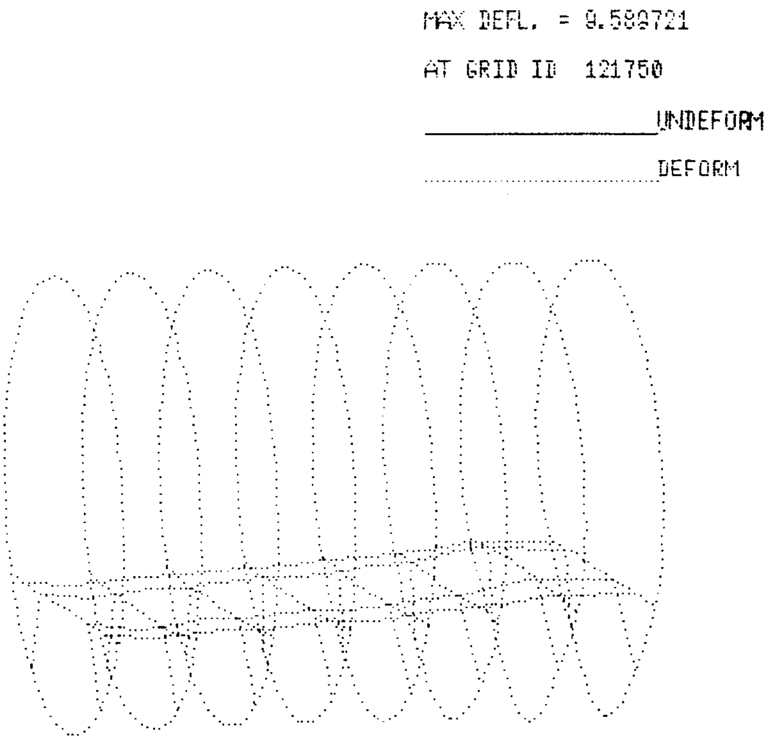


*Figure 4. The Deform Geometry.*

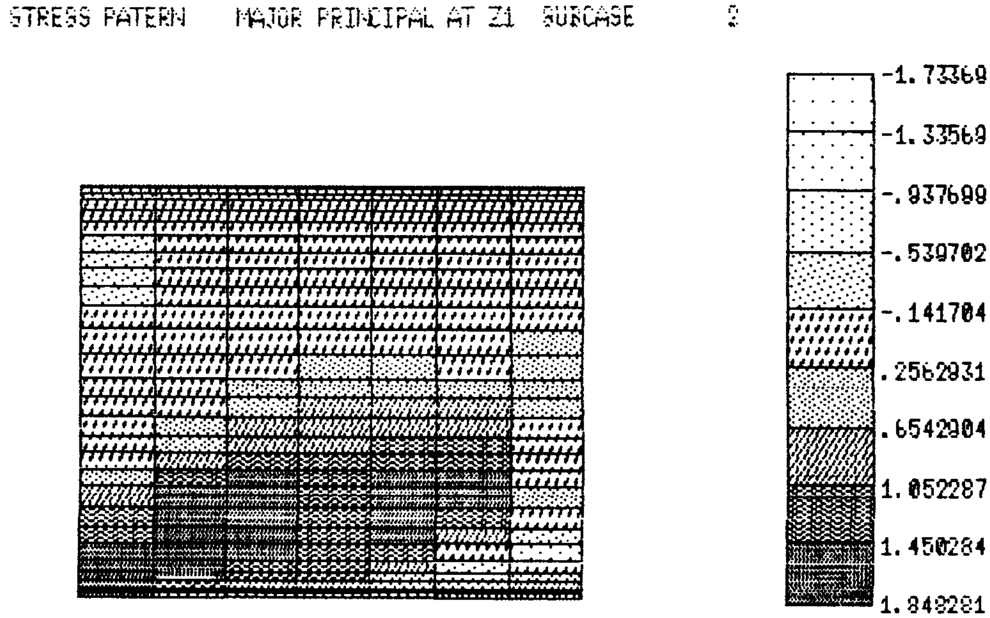


*Figure 5. The Stress Patern.*

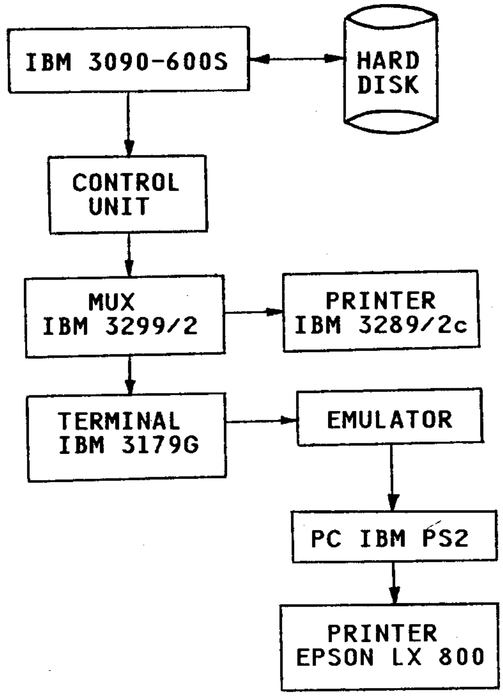


*Figure 6. The Hardware Configuration to operate NUSAP.*