THE BOEING SST PROTOTYPE

INTERNAL LOADS ANALYSIS SYSTEM AND PROCEDURES

ANDREW F. GRISHAM*

THE BOEING COMPANY

This paper covers the management and organization required to implement the Boeing Supersonic Transport Internal Loads Program, the level of technical detail attained via the analysis, the quality control of the final results, and the tools used by engineering to formulate, solve, and display the total program.

* Structures Specialist Engineer

Aerospace Division
Seattle, Washington

1.0 INTRODUCTION

Probably the single most significant change in structural analysis methods in the past 25 years has been the application of matrix structural analysis methods to aircraft and aerospace structures. This change has been possible through the development of the high speed digital computer with large data storage devices, and has been encouraged by the high cost effectiveness of removing unneeded weight from modern commercial aircraft transports. (The Boeing Company considered an expenditure of $150 to $200 per pound of weight saved per airplane to be cost effective during the development of the SST.)

A major consequence of the Boeing Saturn, 747, and SST programs is a significant advancement in the capabilities of structural analysis finite element methods. The structural technical requirements of these programs, culminating with the SST, were such that many millions of dollars and approximately 100 man years of effort were spent in the development of a fully interfaced system of finite element programs to make them handle all types of structural problems, and function efficiently in the process. Furthermore, Boeing developed highly skilled individuals who are very versatile in the use of these tools. More than 40 such specialists were employed within the SST stress group alone during the drawing release phase of the prototypes. The capabilities of these computer programs in type and size of problem treatable; in detail, accuracy and quality control of results; and in applicability to the hardware development, design, and release phases has been well demonstrated in the Boeing 747 and Super Sonic Transport programs.

The reasons for extensive development of a fully interfaced finite element computer program on the SST were: 1) a six-year development period prior to drawing release of the prototypes, 2) a ready access to SST monies, 3) a third-generation computer facility, and 4) a complex environment and structural system. For these reasons advances in the state-of-the-art in finite element analysis were made in: 1) finite element program development, 2) size and scope of problems treated, 3) run time improvement, 4) quality control of solutions, 5) technical procedures, and 6) management of the development and application of finite element methods.

The balance of this paper presents in four sections the background, finite element tools developed, management of finite element program development and use, and examples of the application of these programs to the SST prototype aircraft.

2.0 BACKGROUND

The advent of the supersonic and hypersonic aerospace vehicle together with the continuing demand for higher structural performance, including both weight efficiency and service life, have created the environment for extensive application of the computer to solving structural problems. This situation existed to a high level on the Super Sonic Transport program where the complexity of the structural system, the wide range of load and thermal conditions, and the demands for maximum structural efficiency combined to present a formidable problem to the structures engineers.

To handle a structural system the size of the Super Sonic Transport, in the level of detail required to insure maximum structural efficiencies, required development of a large size computer structural analysis program with a set of fully interfaced peripheral programs that can handle the multitude of supportive tasks that are part of any detail structural analysis.

The requirements set for the structural analysis were stringent, but were management decisions based on extensive experience with such aircraft as the B-70 and the X-20. Specifically, it was decided to insure that there was sufficient size capability that very large problems could be run both as discreet sub-structures and if necessary through an interact system. All six degrees of freedom for each node of a structural element would be included (secondary stresses are very important in fatigue). Structural modeling will include all structural elements (minimize lumping and de-lumping), and must be able to interface all disciplines supporting a structural analysis within the computer structural analysis program.

In addition to these technical requirements there existed one other requirement that may be considered to be the most important. That was a management dedicated to insuring the computer programs were developed, computers were available, and that the engineering drawing release schedules reflected the time it takes to achieve computer developed structural solutions. This management commitment did exist within the Structures Staff on the SST. The result was development of the system of fully interfaced computer programs to be discussed, and perhaps just as important, the tools and techniques used to manage and schedule the data used to support the computer programs and the programs use. The programs to be discussed were applied to the initial release of the Super Sonic Transport structural drawings. The extensive computer use resulted in a very significant weight savings and would have been the major factor in insuring a long service life for the SST structural system.

3.0 FINITE ELEMENT ANALYSIS CAPABILITIES

The primary objective of the SST internal loads program development activity was to produce a set of fully interfaced programs which would, in a timely and cost effective manner, produce a complete set of quality structural element internal loads for each structural element of the SST airframe. Structural geometry, sizing, redundancies, buckling, large deflection effects, loadings (mechanical and thermal), and criteria are accounted for and evaluated in a finite element closed solution. The programs are equally applicable to large and small problems of varying degrees of complexity without imposing significant penalties on either the small or large size problem. Internal loads on each structural element are obtained to avoid time consuming engineering effort and loss of accuracy associated with lumping and de-lumping of mathematical models.

The programs are designed to be fully interfaced: since for all structural analyses it is important that the various computational and display modules at certain points in the analysis procedure report back to the engineering staff before further computation is performed, because the engineering requirements downstream in an analysis change frequently based upon the information learned from the earlier data processing, and because of the large quantity of data which must flow across interfaces without incurring any error in the process. Because of this interfacing it is possible to exercise quality control of the

9 9 all

data processing, and also to conduct engineering reviews of technical and structural aspects of the solutions routinely and as the solutions progress. Built into the interfaced system is a capability for the engineer to bypass any or all pauses in the data processing. These options may be selected where problems are sufficiently small or routine to preclude, in the judgment of those managing the analysis, the necessity of reviewing interface reports prior to the completion of a larger segment of the analysis.

Major objectives in using finite element analyses in support of the SST prototype were:

1) to produce a complete set of quality structural element internal loads for each structural element of the SST airframe,
2) to allow treatment of both thermal induced and mechanically induced loads for all load conditions of interest, including those used for strength design, fatigue, aerodynamic shape and performance,
3) to allow large scale finite element analysis of the entire structural assembly, primary and secondary structure included,
4) to allow fine-grid analyses for stress and deflection prediction within the large scale finite element analysis, using substructuring concepts,
5) to allow three complete internal loads cycles of each structural element prior to 25% drawing release,
6) to produce rapidly a fully interacted solution in a local region where numerous stiffness, grid, and/or connectivity variations are being studied,
7) to allow the inclusion of large deflection and skin post-buckling effects as an integral part of the analysis rather than after the fact,
8) to take advantage of the repeatability and consistency of solution results, which a closed form finite element solution offers, in treating both a single concept and in comparing concepts.
9) to take advantage of a relatively economical tool to treat the large variety of concepts, details, and conditions expected in a technically optimum manner,
10) to support structural development of components, subassemblies, and assemblies by analysis allowing optimization of structural parameters,
11) to allow as detailed an evaluation of the state of stress and deflection in a local region as is required,
12) to produce compatible deflected shapes for components throughout the airframe for load cases of interest,
13) to facilitate definition of those items requiring further structural development,
14) to provide a firm technical base for program decisions on weight, cost, materials, scheduling, and functional reliability,
15) to search for and flag critical stresses for strength, fracture, and fatigue analysis and
16) to support test planning by analysis before~~-the-fact~~ of test loads and support conditions.

Of fundamental importance to the success of a finite element internal loads analysis is the finite element program which generates and solves the stiffness (or flexibility) equations. With an efficient general purpose program of this type the user greatly extends the scope of the problem he can treat. It is necessary to surround such a finite element program with a complete system of fully interfaced peripheral programs if the system is to be fully capable of practically dealing with the full range of complex problems associated with an SST airframe. This finite element system in turn may become a fully interfaced subsystem of the technology staff programs (which include programs used by aerodynamics, performance, stability and control, flutter, and dynamic and static external loads groups).

The SAMECS (Structural Analysis Method for Evaluation of Complex Structures) finite element program system now includes SAMECS and the following fully interfaced peripheral programs: 1) the automatic transformation of loads to the structural grid program, 2) the structural analysis input language program (SAIL), 3) the iterative large displacement program, 4) a library of master dimensions programs of both general and special purpose types which generate geometric data for use by the system as structural geometry becomes more completely defined, 5) a freebody program for forming freebodies within SAMECS models and also for further substructing or updating these models, 6) a substructure interaction capability, 7) automatic input data sorting and updating, 8) an input data format and card sequence logic check program, 9) input and output data display programs, and 10) the post-buckling and diagonal tension effects program.

3.1 SAMECS

The SAMECS program is a powerful tool commonly used to analyze complex structures which may be idealized as an assemblage of plate and beam elements. Other types of elements may be added as necessary via an external stiffness and loads matrix pre-merge option (also used for interaction of substructures). The sub-paragraphs of this section describe the capabilities of SAMECS.

3.1.1 SAMECS Maximum Problem Size

There is no limit on the maximum size problem treatable by SAMECS except, of course, available computer budget. Any structure may be subdivided into any number of substructures. A substructure may contain as many as 2000 six-degree-of-freedom nodes, 10,000 plate elements, and 10,000 beam elements. Loads may be applied in any combination of thermal and mechanical environments up to 40 load cases per pass.

3.1.2 SAMECS Run Time and Computer Costs

Because of the run time improvements incorporated in SAMECS and the large data storage capabilities of the Boeing computing facilities, SAMECS is currently considered to be capable of processing large structural systems and with less run time and computer cost than any other known finite element program. For a structure which fits within the size restrictions of the SAMECS substructure (and therefore does not require interaction to complete the analysis) the computer cost varies between 1/2 cent and $2\frac{1}{4}$ cents per node freedom per load case with the unit cost decreasing as the complexity of the problem decreases and as the number of load cases processed increases. Data processing costs are slightly less when SAMECS runs are processed using the full core of the computer in a single job rather than multi-job processing mode. However, because full core jobs must be scheduled, smaller problems are usually processed in the multi-job processing mode in order to obtain fast turn-around.

Structures which do not fit within the size restrictions of a SAMECS substructure and must be interacted cost between 1/2 cent and 6 cents per node freedom per load case with the unit cost decreasing as the complexity of the problem decreases, as the number of load cases processed increases, and where

3.1.2 (Continued)

more than one interaction run is required. These cost figures are based on a computing facility cost of $2.75 per Computer Resource Unit.

Computer data processing run time can be approximately computed from run cost since the Boeing CDC 6600 facility costs approximately 250 computer resource units or $687.50 per hour for SAMECS full core (stand alone) processing. No prediction of run time is possible in the multi-processing mode since the occupancy on the computer is then a function of the number and types of other jobs processing simultaneously, relative job priority in the system, and utilization of disk, tape, printers, core, card readers, card punches, and other components of the computer by the multi-processing jobs. However, a typical statistic on a 240-node six-degree-of-freedom job having six load cases would be 55 computer-resource-units and an occupancy on the computer in the multi-processing mode of 18 minutes.

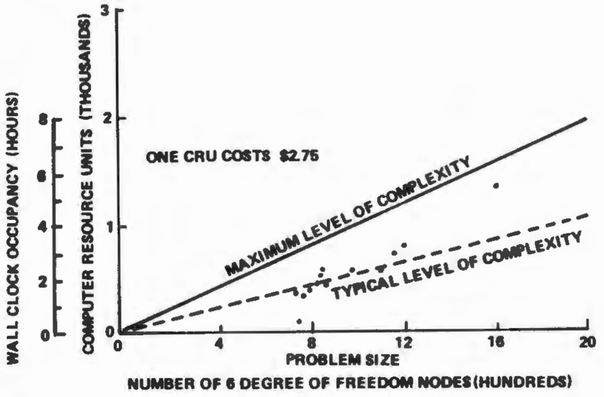

Figure 3.1.2: SAMECS INTERNAL LOADS RUN TIME AND COST

On the SST program 800 to 1000 node runs processed routinely overnight. The cost of processing a 1000-node (6000-degree-of-freedom) run with 20 load cases is typically 500 computer-resource-units and the occupancy full core is typically 2 hours. A graph showing typical computer data processing statistics on run time and cost is shown in Figure 3.1.2.

3.1.3 SAMECS Finite Element Capabilities

Nodes, plates, and beams may be randomly numbered in SAMECS. Node freedoms may be fixed, free, specified or sprung.

Plate elements are isotropic, orthotropic, isotropic stiffened, or orthotropic stiffened; may be pre-strained, heated through their thickness and across their surface linearly, may be offset, and allow heating of offsets. The quadrilateral plate element is always in precise force and moment equilibrium, even though warped, allowing moment and force equilibrium checks for all structural assemblies of elements without exception. All varieties of plate elements posses both inplane shear and stretch, and out-of-plane shear and bending. The quadrilateral plate element is formed through the use of a centrally located fifth node and four triangular plate elements. An inplane rotational stiffness for the plate element is generated to assist the user in generating three independent rotational stiffnesses at all plate nodes. The effect of this inplane rotational stiffness on plate element inplane translational stiffness is not allowed to exceed 1% for a plate element regardless of aspect ratio.

Beam elements are curved or straight, may be heated axially and through their depths along both major and minor axes of bending, may be pinned torsionally and about both bending axes at each end of the beam, may be offset in any orientation relative to its end nodes (same as the plate element), allows heating of offsets, has constant, linearly varying and piecewise linearly varying section properties, and may be loaded between nodes with uniform, linearly varying, piecewise linearly varying, and concentrated loads in both local and general coordinate systems. Section properties and loads may be defined at as many as 50 stations between the nodes of a beam element.

In addition to plate and beam elements, super elements (stiffness matrices or reduced stiffness matrices for other structures) and solid element stiffness and loads matrices (full or reduced) may be included in a SAMECS analysis via a pre-merge feature.

3.1.4 SAMECS Solution Capabilities

Both the multi-processing and the full core version of SAMECS will produce: 1) internal loads, stresses, deflections, and reactions, 2) reduced stiffness matrices up to 1000 x 1000 for a substructure or interacted group of substructures, 3) reduced flexibility matrices up to 1000 x 1000 for a substructure or interacted group of substructures, 4) interaction of substructures, and 5) pre-merge, which allows substructure interaction within SAMECS.

Equilibrium checks comparing the equilibrium of input loads and resulting reactions, conditioning checks on every freedom of the stiffness matrix, stiffness matrix maps, run diagnostics, and timing information for each phase of the SAMECS solution are reported automatically for all runs.

SAMECS uses the SOLPAC (SOLution PACkage) solution program based on Choleski decomposition.

3.1.5 SAMECS Output Communication

SAMECS outputs all of the data described in Section 3.1.4 in both printed form and on multi-file output tapes. In addition to this data, all input data, element transformation, stiffness, and loads matrices, and gross stiffness matrices are available on request. The multi-file output tapes are used for communication with the SAMECS peripheral programs.

3.2 SAMECS PERIPHERAL PROGRAMS

A system of peripheral programs was developed to interface with SAMECS. The purpose of these programs is to facilitate the preparation of mathematical models, the interpretation and application of the results, to add certain analysis capabilities of an iterative nature, and to permit interaction analysis using substructures. The SAMECS peripheral programs are therefore capable of dealing with any problem which the SAMECS program can treat.

Interfaced programs were developed for use on the SST for several reasons; 1) there is no overhead associated with an executive system, 2) the CDC 6600 system has in effect already built in an executive control system which allows the stringing together through the use of control cards one or more of the SAMECS system programs, 3) checkpoint restart options are automatically available when interfaced programs are used, 4) problems which may develop in one program do not effect any of the other programs, 5) a string of activities may be conveniently grouped together to make a computer run of reasonable run time depending upon the urgency of the problem, availability of computer resources, and confidence of the analyst, 6) it is logical to break programs where engineering checks and/or decisions routinely occur, 7) new programs having increased capabilities are used in conjunction with SAMECS and all of the SAMECS peripherals by simply complying with system interface requirements. It is pointed out that some of the above points may be equally applicable to integrated (or executive system controlled) programs. A chart showing the flow of data through the SAMECS system of programs is shown in Figure 3.2.

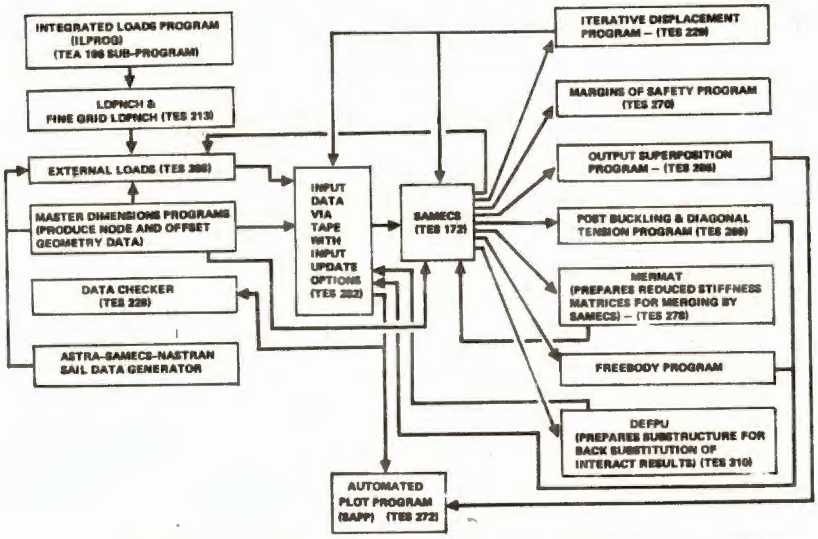

Figure 3.2: THE FULLY INTERFACED SYSTEM OF SAMECS FINITE ELEMENT PERIPHERAL PROGRAMS USED BY THE SST STRESS GROUP

3.2.1 SAMECS External Loads Transformation Peripheral Program

The SAMECS External Loads Transformation Program completely eliminates the need for the engineer to manually prepare nodal loads data for SAMEC runs. It furthermore provides a means of positively guaranteeing the quality of the nodal loads data to management prior to a SAMECS run, and it provides complete reports on the formulation of the loads data so that future questions regarding the source of loads data are easily and completely answered.

The SAMECS external loads program performs a multiplicity of tasks. These tasks are accomplished in four distinct sections of the program. These four sections were used to prepare virtually all of the loads data for SST mathematical models. The sections: 1) transform local loads of all types to the structural grid (including mechanical, pressure, and inertia type loads), 2) distribute loads outside of the domain of mathematical models to the nodes on the boundaries of the models using engineering theory, 3) perform various types of shear and moment calculations using the node loads computed in 1) and 2) as the source of data for the integrations, and 4) scale unit load conditions to form actual load conditions, convert loads into symmetric and anti-symmetric loads where structural symmetry is taken advantage of, combine, scale, and/or correct unit load conditions, and merge reduced loads from substructures for substructure interaction with SAMECS. This program is fully interfaced with loads group programs for automated data transfer of actual loads scalers. A more detailed description of these capabilities is given in Section 3.2.1.1 through 3.2.1.4.

3.2.1.1 External Loads Peripheral Program Transformation of Local Loads Option

The transformation of local loads option is of primary importance in reducing the engineering effort required to prepare loads input to SAMECS. Input consists of the SAMECS node coordinates and a list of nodes, beams, and triangular and/or quadrilateral plates which are loaded. Loads may be applied as forces (force vectors or moment vectors in space), pressure acting on panels, pressure acting on projected areas of panels, and loads acting on beams. The program: 1) equates a uniform pressure load (pressure may be specified as other than normal to the surface and to act on the true or projected area of the panel), on a triangular or quadrilateral (may be warped without loss of equilibrium) panel to an equivalent system of forces at its corners, or at any other group of points (not necessarily the corners of the panels); 2) equates beam loads and point loads and moments (vector direction of loads may or may not be required depending upon option selected) to an equivalent system of forces at the ends of the loaded elements or at any other group of points; 3) performs the same operations as described in 1) and 2) except computes weights of elements from weights data supplied by the weights group and then computes for each of the rigid body unit accelerations the equivalent inertia forces induced on the corner (or other) node points; 4) sums all element corner (or other) forces at nodes for each load or unit inertia conditions automatically and stores results with an identifying unit load case number, 5) performs equilibrium checks of all load and unit acceleration conditions formed (including necessary summations for calculation of centers of gravity of mass groups), 6) calculates for the air loads group the projected areas and centroids of projected areas of air loads group panels on the three-dimensional surface.

At early stages of structural development, finite element analysis air loads are usually received from the air loads group as generated on a two-dimensional aero-elastic grid balanced by vertical and pitching accelerations. At more advanced stages forces and accelerations in all directions are of interest, including the three-dimensional slope of all aerodynamic surfaces. Achieving a balance for the two-dimensional problem is straight forward since the two-dimensional aero-elastic panels are overlaid on the structural panels and using the "LINES AND PLANES" program precise transfer of the air paneling systems pressure load to the structural paneling system and integration is accomplished. However, when a three-dimensional balance is undertaken, then the air loads group must be given the precise centroids and projected area of the structural panels which are equivalent to the aeroelastic paneling system. In this way drag and lateral forces and the effect of surface slope is taken into account in the balance, and the accuracy of the total effort is assured. The transformation of local loads option produces the required information on punched cards for the air loads group in a format compatible with the balance programs.

The transformation of local loads capability is the key to balanced load conditions, since it permits exact transformation of loads without incurring any of the engineering effort that would otherwise be associated with such precision.

3.2.1.2 External Loads Peripheral Program Transformation of Loads to Model Boundaries Option

The boundary loads option is used to apply loads to mathematical model boundaries prior to the time when an interact is accomplished which encompasses that boundary in a larger solution, or for problems where the inaccuracies of load distribution induced by assuming simple engineering theory are considered to be small either because of the nature of the structural system, or because the boundaries are sufficiently removed from the region of interest within the model. Loads which are applied to any reference point in space are in turn applied equivalently to all of the nodes in a defined section made up of plates and beams (which may or may not be offset). In addition to calculating the equivalent node loads on the section and performing appropriate equilibrium checks on the resulting set for comparisons with the orginally applied loads, this option also prints out section properties and resulting stress distributions in the boundary elements.

Elements need not be oriented perpendicularly to the section or boundary plane since in aircraft structure such an assumption will in general cause an ovalization of the section and different frame bending moments and skin shear flows.

3.2.1.3 External Loads Peripheral Program Shear and Moment Option

In addition to the equilibrium checks which are performed routinely on all load conditions formed, it is sometimes desirable to perform a stepwise integration through one or more load cases, transforming the resulting forces and moments to specific points of interest. A series of integration intervals may be defined using ranges of coordinates, a list of nodes, or a combination, and integration may be obtained approaching the SAMECS model array of nodes from different directions. It is the purpose of the Shear and Moment Option to provide a general tool for obtaining the specific information needed to produce checks and plots of data not fully covered by the automatic equilibrium checks.

3.2.1.4 External Loads Peripheral Program Unit - Actual Loads Options

The heart of the external loads program is the unit actual loads routine. This routine accommodates new load conditions and changes to old load conditions on short notice, and gives the user the check information required to verify the quality of all loads data. This module takes the unit conditions formed by the previous options, modifies them as required, forms new unit conditions from existing unit conditions, adds any special unit load conditions, receives unit-actual load condition scalars on punched cards from the loads groups and scales and adds the unit load conditions to form actual load conditions, breaks the load conditions into symmetric and anti-symmetric components, performs equilibrium checks of actual load conditions about selected points of interest, and performs the merging of reduced loads from several substructures into a single set of loads for an interact run. The resulting node loads card images are sorted in a proper sequence for use by SAMECS and written on tape ready for updating onto the SAMECS input data tape. When interaction between substructures is being accomplished and the node and/or load case numbering differs from that being used in the interaction solution, correspondence tables are read by this option and the revised numbering automatically assigned to all nodes and load conditions.

3.2.2 SAMECS SAIL (Structural Analysis Input Language) Peripheral Program

The SAIL program is a structural analysis input language which allows the user to take advantage of repetition in the numbering, sizing, and geometry in the preparation of the structural elements comprising a mathematical model. Node, plate, beam, and some loads data may be generated using SAIL. Equations may be written for, or tables input directly in, the SAIL program to assign values to the various required input parameters of SAMECS. The SAIL program is particularly useful for generating super refined grids where repetition in the mathematical modeling is prevalent. Models constructed in part or in total with SAIL may be modified in ways that result in completely new data sets but require only small changes to the original SAIL data. For example, quadrupling the refinement of a mathematical model grid may require only a one card change in SAIL, but it may result in hundreds of new card images being produced on tape for SAMECS input. The SAIL program is also capable of outputting data decks for ASTRA and NASTRAN finite element programs using a SAIL input deck that orginally produced a SAMECS data set.

3.2.3 SAMECS Iterative Displacement Peripheral Program

The iterative displacement program is used in conjunction with SAMECS to correct equilibrium equations of any finite element mathematical model for the effects of large displacements. The program has been used successfully to treat a large number of structures for which the rotations of the elements due to deflections induced by loads and temperatures were important in increasing or decreasing the stresses in the finite elements. Included in the list of analyses performed recently using the method are: 1) the 747 fiberglass honeycomb wing leading edge panels, 2) the SST cab acrylic window analysis, 3) several analyses of skin-stringer, integrally stiffened, and sandwich panels, and 4) a beam-column type improved stress analysis in the aft body of the SST immediately aft of the rear spar. Analysis results have been substantiated by test for the 747 fiberglass honeycomb wing leading edge panels and the SST cab acrylic window.

3.2.4 SAMECS Node and Offset Data Generation Programs

The geometry of SAMECS mathematical models is entirely defined by node coordinates and element offsets from the nodes. This data is generated in many ways, occasionally manually, but more frequently by SAIL, one of the special purpose geometry generators (such as that for windows), or finally and most frequently by a library of general purpose geometry programs developed and used by a project support group called Master Dimensions.

SAIL is used where the shape of the model being treated may be adequately described using mathematical equations for the surfaces, planes, lines, and intersection points involved. The engineer codes these equations directly in SAIL language and can have the resulting nodes and offsets back the next day. Time-sharing teletype terminals are frequently used for small programs to allow immediate results in support of a modeling effort.

Specialized programs have been written by Master Dimensions and turned over to the structures groups for their use. The Master Dimensions "window" program is a good example. The window program allows the user to stipulate: the thickness and corner radii of the several panes of glass, the thickness of the rebate, the thickness of the seal material around the edge of the glass, post and sill geometry (where the post and sill is modeled as a built up section of plates and beams), hard points such as intersection points of posts with sills and stiffeners with posts and sills, and finally the fineness of the grid into which the window panes are to be divided. From this information the Window Master Dimensions program generates node coordinates for the glass, seal, posts and sills assembly. All coordinates may be skewed to conform precisely to the true geometry of the vehicle and facilitate interaction of window with surround and/or imposition of surround displacements on the post and sill structure. Because a Master Dimensions program is used to generate this node data, it is known: that the seal springs which transmit shear and moment to the edge of the glass panes are precisely normal to the plane of the glass, that the glass pane is precisely flat, and that the loading which is dependent upon the node coordinates is accurately and precisely applied. Small inaccuracies in the node coordinates of the window analysis could result in large errors in internal loads and displacements. Similar programs are used to obtain refined grids in the corners of cutouts.

As the project definition of hardware to be analyzed evolves then the Master Dimensions group geometry definition which is used for preliminary lofting, firm lofting, drawing release and manufacturing can be referenced to obtain node and offset data for the structural analysis. Since the primary concern of the finite element structures analyst is not a tight tolerance on the accuracy of the geometric definition (a .1" difference in the radius of the fuselage monocoque will cause no significant change in internal loads) but a precisely smooth definition (a .1" joggle in the surface of the fuselage monocoque over a .5" arc length can ruin the local results, is difficult to detect, and costly to correct). Master Dimensions is used almost exclusively by structure analysts performing medium and large size analyses using SAMECS. Data supplied by Master Dimensions to the analyst is node coordinates on magnetic tape or punched cards, surface normals on magnetic tape, and direction cosines of the lines of intersection of structural element intersections such as frame web with stringer web, again on magnetic tape.

Offset data is generated by an offsets program using SAMECS structural data, section y bars, and the magnetic tapes supplied by Master Dimensions. Some additional comments on the function of Master Dimensions are made in Section 4.3.1 dealing with quality control of the mathematical analyses.

3.2.5 SAMECS Freebody **Peripheral Program**

The Freebody Program acts as a data extractor in that it extracts from any SAMECS solution a selected subset of the data which constitutes a complete SAMECS data set within itself. Deflections solved for in the previous SAMECS solution may be automatically imposed on the boundaries of the subset structure. Reactions obtained from the subset SAMECS run are the freebody loads. Furthermore, the new input data tape formed by the Freebody program may now be altered in sizing, refinement, heating, geometry, and/or loading and with the reactions applied as loads the subset may be reprocessed for a SAMECS solution, or the altered input data tape may be used to update the original SAMECS input data tape so that the most current information in that particular region of the structural system will be included in the next run of the original model.

3.2.6 SAMECS Substructure Interaction Programs, Including DEFPU and MERMAT

A capability to substructure (or partition) the mathematical model of a complex structural system was of key importance to the SST structures staff in obtaining adequate internal loads analyses simultaneously with the development and drawing release of the SST prototypes. Substructuring and substructure interaction was developed for several reasons:

1) Use of substructures to idealize structural systems permits a detailed idealization within each substructure at the same time a comprehensive analysis of the total structure is obtained.
2) Structure not well defined may be modeled, using a coarse grid idealization, and used to accomplish, as desired, interaction with fine grid substructures for purposes of improving the boundary conditions of the fine grid models.

3) All information required to idealize a large section of structure may not be available at the time an analysis of some portion of that section is required. Use of substructures permits setup of each portion as design information comes available and also allows interaction at a later date with the whole structural system.
4) Substructure analysis reduces the waiting period for large size idealizations, setup, and execution, and spreads out the model preparation and data reduction effort more uniformly over the design and drawing release periods.
5) Failsafe analysis is economically accomplished using the substructure interaction approach.
6) Computer program modules may be designed to handle a certain fixed size substructure which is primarily a function of the computer system used, and run times may be appropriately optimized around that size without affecting the generality of the type, size, and treatment of structures problems by the analyst.
7) Run times on computer equipment may be broken into several segments of controllable size, and with engineering check points for quality control between the segments: to allow verification of analysis progress and quality, to eliminate the possibility that a very large block of computer processing time may be lost due to computer equipment malfunction, and to allow engineering access to valuable internal loads and deflection data prior to obtaining the fully interacted final result.
8) The flexibility of an interact system, properly designed, is such that literally hundreds of alternative routes through the interaction procedure are available. This provides an analysis system which has the best opportunity of satisfying each technical and scheduling problem in an optimum manner. For example, options are available as to: sequence of data processing of the substructures, whether only stiffness or stiffness and load interaction is required, which of the substructures will be back-substituted from boundary deflections to obtain internal loads and deflections and for what load cases, which substructures will not require reduced stiffness matrix updating in a given cycle, whether a full or a partial interaction is required, whether a pure interaction or a hybrid interaction is required (a hybrid interaction is one in which certain elements are included immediately in the interaction run and internal loads and deflections are obtained directly from the interact run for these elements), whether fail safe or cracked structure considerations should alter the interaction procedure, and whether first, second, third, etc. level interactions are to be employed.

Interaction of substructures is accomplished through the use of four key programs: SAMECS, External Loads Peripheral, MERMAT, and DEFPU.

SAMECS pre-processes each substructure to obtain a reduced stiffness and a reduced loads matrix. MERMAT merges the reduced stiffness matrices from the several substructures using a correspondence table for nodes. The External Loads Peripheral Program merges the reduced loads matrices from the several substructures using the same node correspondence table and a load case correspondence table. SAMECS reads the merged matrices and solves for the interacted boundary displacements. DEFPU imposes the interact displacements on the boundaries of each substructure using the same correspondence tables used above, and SAMECS is used to calculate internal loads and deflections within each substructure.

The MERMAT program has the capability of transforming stiffness matrices to simulate the effects of pivoting and/or translation of a substructure relative to other substructures, or to accommodate interaction of substructures having different coordinate systems.

3.2.7 SAMECS Input Data Sort and UPDATE Peripheral Program

The UPDATE program eliminates the need for all deck sorting, sequencing, and stacking for SAMECS runs. All cards or card images on tape produced by any of the peripheral programs to SAMECS have entered in card columns 73 through 80 a uniquely defined sequence number. These numbers are also carried on all of the standard input forms used in preparing SAMECS input data manually. This system together with the total elimination of node loads greatly reduces the chance of engineering error in forming, making additions, corrections, or deletions to SAMECS data sets on tape.

In addition to sorting, adding, correcting, and deleting SAMECS data sets, the UPDATE program automatically provides and inserts all blank cards, data delimiters, and standard header cards with sequencing information entered for the user.

3.2.8 SAMECS Data Checker Peripheral Program

The Data Checker Program is used to check the input data before a SAMECS run is attempted. If the SAMECS data set is accepted by the data checker, then the successful execution of that data set using SAMECS is guaranteed, and it is almost certain that scheduled computer time for the run will be efficiently and profitably used. It checks for the logical sequence of SAMECS input data sections (including nodes, plates, beams, and loads) and for the logical sequence within each section. The sequence numbering in columns 73 through 80 is checked along with the format of each field of each data card. The program will then simulate the steps which SAMECS performs in reading in the data, screening it, and computing element matrices. It produces the identical diagnostics that SAMECS would produce, but does not generate the matrices. This requires 20 to 25% of the residency processing time normally required by SAMECS in the SAMECS generate phase saving a small amount of computer time, but more important, because of its small core requirements, rapid turn-around is assured.

3.2.9 SAMECS Automatic Plot Peripheral Program

A major contributor to the quality of SAMECS analyses is the automatic plot program. Although the Master Dimensions group performs numerous plots of the geometric data which that group generates for SAMECS, the final check on node geometry and the element orientation and connectivity to that node geometry must be made using exclusively the SAMECS input data deck. Equally important is the capability to display the solution, observing trends in load paths, insuring that there are no undetected "soft spots" in the analysis, and finally for use by the strength, fatigue, fracture, mechanical systems, loads, performance, and design groups for displaying specific information in support of their respective problems.

The SAMECS Automatic Plot program will produce any number of two-dimensional and three-dimensional views constructed from SAMECS input data or output results, or from SAMECS External Loads Transformation Program Elements. The whole structure or any parts thereof may be plotted. Two-dimensional and three-dimensional views of models are plotted using the Cal Comp Plotter. The program also plots internal loads, stresses, and node deflections in any of the standard or three-dimensional views desired.

Features available are: arbitrary angular orientation and scale of system axes in the displays, automatic compression of plot size to fit a page dimension specified, labeling of nodes, plates, beams, element local coordinate system orientations, arbitrary lettering sizes, labeling of any and all internal loads data produced as output by SAMECS, and construction of three-dimensional deflection plots. Data produced for the SAMECS Loads Program, for and by SAMECS, and by the SAMECS Output Superposition Program may be plotted.

3.2.10 SAMECS Output Superposition Peripheral Program

The output superposition program allows superposition of unit or actual load cases processed by SAMECS using appropriate scalars. Plate and beam element stresses and internal loads, and node deflections and reactions may all be superposed in the same run as required. Different loadcases in the same SAMECS run, or same or different loadcases in different SAMECS runs of the same structure may be superposed. The program is commonly used to combine critical design flight conditions with appropriate pressures and thermal environments, to form left-and right-side results from symmetric and anti-symmetric results, to combine load conditions appropriately for use with the post-buckling peripheral program, to ratio loads from limit to ultimate, and to perform max-min critical condition searches of the superposed data. Max-min data is printed at the bottom of each element's superposed load conditions, and also by itself as a max-min envelope package. Output data on tape and in printout is in precisely the same format as SAMECS output data, so that peripheral programs may operate with the superposed data as well as the original SAMECS data.

The Output Superposition Program is frequently used to prepare a special set of data required by a specific group, eliminating those conditions not of particular interest. In this way the user is able to focus quickly on the pertinent data for his problem, and the amount of printout data retained in each area is minimized. The SAMECS Margins of Safety Peripheral Program is even more selective in the superposition and display of results.

3.2.11 SAMECS Margins of Safety Peripheral Program

The Margins of Safety program is used for detecting and displaying critical stresses in SAMECS beam and plate elements. This program is capable of: 1) extracting internal loads and section properties for selected beam and plate elements from one or more SAMECS output tapes dealing with the same structure, 2) preparing design loads by superposing the SAMECS internal loads using specified scalars, 3) selecting the proper allowable stress for each design case corresponding to the thermal condition included with that design case, 4) calculating the fiber stresses and margins of safety in the extreme fibers of the selected beams, 5) calculating plate element principal stresses and margins of safety for selected conditions, 6) analysis of beam elements with section properties different from those used in the last SAMECS solution, 7) presenting the input and output data with various printout options in an easy-to-read format, 8) x-y plotting of beam unit moments, axial loads and shears as ordinates, and beam numbers as abscissas, 9) x-y plotting of beam section properties, design loads, and stresses, and 10) execution of several sets of beams and plates in one run.

3.2.12 SAMECS Post-Buckling (Including Diagonal Tension) Peripheral Program

For structural systems where the buckling of the structure under load causes major redistributions of internal loads the SAMECS Post-Buckling program is most important. There is little point in producing internal loads and deflections for a complex structural system by a detailed finite element analysis procedure and then leaving the engineer to his own devices to correct the solution for buckling effects which when incorporated alter the solution or even change the signs of the internal loads in many of the structural elements.

The Post-Buckling program treats post-buckling effects (including diagonal tension) in SAMECS plate elements, using the constant initial strain (ϵ_x, ϵ_y, γ_{xy}) inputs available for SAMECS plate elements, in an iterative procedure. The program evaluates each plate element's degree of buckling for each load case and determines the corresponding initial strains for input to SAMECS. It also prepares symmetric and anti-symmetric pre-strains where principles of structural symmetry are taken advantage of in the modeling. Following are the important aspects of the program:

1) Post-buckling can be handled by a series of SAMECS iterative runs using the initial strain capability for each run. Iteration gives the model opportunity to redetermine load paths due to compression buckling and diagonal tension effects, and to re-evaluate the buckled state of plates in as many cycles as are required to achieve the desired degree of convergence. A closed solution is obtained which exactly meets force equilibrium and deformation compatibility in the model since the SAMECS program is used to process the pre-strain plate data.
2) Compression buckling in one direction is permitted without affecting the stiffness in the other direction.
3) Compression buckling in both directions is permitted with different stiffnesses, respectively, which correspond to the degree of buckling.
4) Shear buckling is permitted, which produces diagonal tension in plates.
5) Combination of compression and shear buckling with tension present is permitted.
6) No change in the stiffness matrix of the structure is required in the SAMECS program. Consequently, all load cases may be treated simultaneously in a single series of iterations.

3.3 PROGRAMS INTERFACING WITH THE SAMECS SYSTEM OF PROGRAMS

A large number of programs too numerous to list or discuss here, are interfaced with the SAMECS system of finite element programs. They encompass all of the technology areas outside of the Stress group, and include static and dynamic loads, flutter, stability and control, performance, and aerodynamics groups. Some of the programs provide data directly to the SAMECS system. Such data might be aeroelastic panel geometry, panel pressures, load factors, powerplant thrust, etc. Other of the programs may require data from the SAMECS system such as deflections, internal loads, reduced flexibility matrices, and reduced stiffness matrices. Several eigenvalue programs, dynamic response, transient response, aeroelastic, and stability and control programs may at the option of the analyst use SAMECS originated data. The SAMECS system multi-file concept embraces a generality which encompasses or can be easily made to encompass all of the technical programs which would have reason to use SAMECS originated data. Furthermore, as programs are developed or improved within the various technology areas, there is sufficient documentation available on the SAMECS system to allow these programs to maintain a fully interfaced condition with the SAMECS system where applicable. Figure 3.3 shows technology group computer programs and the flow of data between groups across program interfaces.

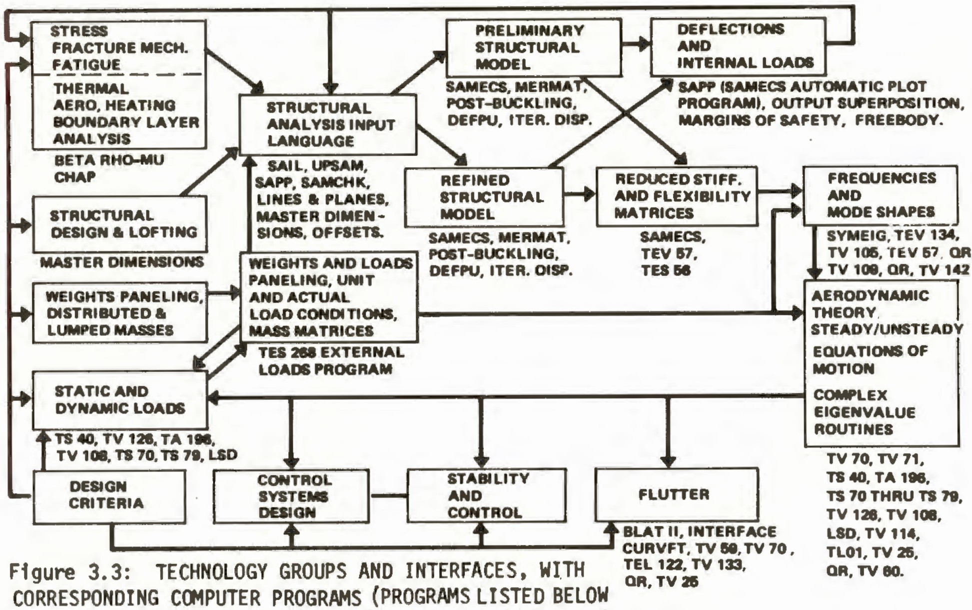

Figure 3.3: TECHNOLOGY GROUPS AND INTERFACES, WITH CORRESPONDING COMPUTER PROGRAMS (PROGRAMS LISTED BELOW BOXES ARE USED TO ACCOMPLISH ACTIVITIES INSIDE BOXES)

4.0 MANAGEMENT OF FINITE ELEMENT PROGRAM USE

The attention to detail required of management and engineering to produce analyses of the behavior of complex structural systems has become more demanding as the scope and level of detail of the analyses has been extended. Members of Boeing Structures Management recognize that the solution of the algebraic equations for the structural internal loads system and the peripheral processing associated with preparation and reduction of the analysis is only one part of the total task. Important to the success of the analysis effort is the involvement of engineering managers who have a thorough understanding of the tools being used and their application to the design task, and who recognize that only with proper emphasis on the disciplined development and management of the total problem are we able to use the computer to the extent that the most up-to-date computing facilities allow. Without a well coordinated and thoroughly integrated effort with a specified end product deliverable on a committed and realistic schedule, the results obtained may lose much, if not all, of their value.

A set of management and technical procedures were developed at Boeing for the SST program and routinely employed in treating the SST structure. Standardized procedures for construction of the finite element mathematical models and for operation of the finite element programs were adopted. Documentation and standardization of all aspects of the operating procedures for the SAMECS system was developed and completed. By following these procedures Boeing was able to assure consistency throughout all phases of solution, proper record keeping of all activities preparing finite element models, timely and efficient coordination of model construction and associated data transfer, quality control of solution results, and complete documentation of analysis results to give a complete technical historical perspective to design decisions.

Management procedures adopted relate to: 1) preparation prior to undertaking the finite element analysis effort, 2) organization required to implement modeling activity, 3) execution of the solutions with thorough quality control of all facets of the analysis effort and 4) data transfer of the results to the technology groups relying on the finite element information. Each of these aspects of the management and technical procedures are discussed in Section 4.1 through 4.4. A brief discussion of the scope and cost effectiveness of finite element methods for the SST is presented in Sections 4.5 and 4.6.

4.1 PREPARATION PRIOR TO UNDERTAKING THE FINITE ELEMENT ANALYSIS EFFORT

Management preparation includes:
1) Adoption of a Finite Element modeling philosophy,
2) Establishment of a mode of management control,
3) Standardization of technical procedures, and
4) Adoption of modes of information transfer between Technology Computer Programs.

4.1.1 The Finite Element Modeling Philosophy Used on the SST

Management adopted an overall finite element modeling philosophy for use on the SST program. This general philosophy was adopted to provide guidelines for construction of models, and was modified as necessary to fit the objectives and timing of the various phases of the SST prototype development and drawing release activities. The modeling philosophy adopted stated that:
1) Substructuring of the structure would be required,
2) Six-degrees-of-freedom would be used at all nodes in the finite element grids,
3) Lumping or delumping of primary structural elements would be avoided, every physical structural element would be modeled with corresponding finite elements adequately representing the discreteness, stiffness, and connectivity of the actual structure.

This philosophy ground rule regarding lumping and delumping merits further comment. Lumping and delumping of structural elements were avoided where possible because: such activity is time consuming, it acts to degrade the quality of the mathematical model, and it makes it more difficult to later use the plate element post-buckling and diagonal tension program. (This is so because the buckling loads are a function of the geometry of the plate elements or element groups and if the finite element plate isn't a reasonably good representation of the shape of the actual plate then special parameters have to be prepared for the post-buckling program furnishing this information for each plate which is different. If there is good correspondence, then the handling of the plate elements to be buckled is automatically accomplished by the computer program from the basic input data.)

It is important to note that only because of the large problem size capabilities and improved efficiencies of the SAMECS program were we able to avoid lumping. SAMECS was applied using this philosophy on virtually all of the SST wing and body primary structure.
4) Key section properties of stiffener elements will be modeled whether it appears that they are significant to the analysis or not, i.e., A, I_x and I_y, and j, will be included in beam elements.
5) Additional subelement refinements will be used in areas of rapidly varying stress gradients to produce the required results, for stress, fracture and fatigue groups.
6) Only balanced load conditions will be analyzed.
7) Master Dimensions will be the source for geometric data for fine and superfine grid finite element models.

8) Engineering will follow a strict set of technical and management procedures. Specific quality control checks will be made on all analysis activity, specific minimum standards of quality will be met, and summary reports on quality will be published for all phases of modeling activities.
10) Non-linear, large displacement, and post-buckling and diagonal tension capabilities will be used at the finite element level to answer as required the questions of effectivity and overall stiffness of structural elements and models.
11) Joint stress severity factor analysis will require the modeling of discrete fasteners in interacted or partially interacted analyses to allow optimum joint design.

4.1.2 Established Manner and Modes of Management Control of Finite Element Modeling

Management decisions were made at the inception of all internal loads modeling activities in support of SST structural analysis efforts. These decisions related to the following aspects of the problem: 1) technical interfaces, 2) sign conventions, 3) coordinate systems, 4) adoption of standard input forms, 5) adoption of quality control measure, 6) adoption of procedures, 7) implementation of the system by which management would review modeling activities to insure compliance with their requirements and procedures, 8) definition of items needing management approval before committing runs, 9) definition of the level of detail, type of forms, and cataloging system to be used for recording all math model inputs inlcuding geometry, sizing, loads, and stress calculations in support of model development, 10) assignment of math models to specific stress group engineers who would be responsible for the structural input to the models, assembly, loading, quality assurance, selection of output display, overall coordination of interactions of substructures, overall coordination of data transfer between technology groups, direction of computer programming, and consultation on finite element modeling in total, and 11) assignments for scheduling of the major milestones finite element schedule, determination of manpower requirements, and determination of computer budget and monthly use required to support the major milestones schedules.

Where management procedures adopted for the SST called for specific management approval, the SAMECS system was designed to offer management optimum visibility of key steps taken by the computer, and quality control check sheets were designed to make it impossible for engineering to continue beyond certain points without recognizing the management requirement for approval.

Finally, SST management provided data storage and display areas to provide maximum management and technical staff visibility of the analysis efforts at all stages of development. In these areas were kept math model node and element diagrams, interact, weights, and air loads paneling diagrams, major milestones schedules, computer data management charts, input and output computer data logs, tapes, cards, printouts, plots, and geometry, stiffness, external loads, and structural sizing references.

4.1.3 Standardized Technical Procedures for Construction of Finite Element Models

Standardized procedures for engineering to follow in the construction of SAMECS models and for the operation of SAMECS and the peripheral programs to SAMECS were developed for use on the SST program. These procedures were adopted to assure that all associated with the definition, scheduling, preparation, and application of mathematical models were fully aware of the system needs in insuring quality, compatibility, proper utilization, and adequate documentation. Such procedures were used to acquaint new employees and subcontractor employees with the requirements of using the SAMECS system. Topics discussed in the procedures include ground rules, peripheral programs, technology interface requirements, input requirements, substructure interface requirements, descriptions of specific coordination which must be accomplished on interfaces to assure timely and proper support, descriptions of the modeling philosophy to follow, general comments (including a list of common pitfalls to be avoided), and an item by item description of each of the major milestones on the major milestones schedule.

4.1.4 Information Transfer Between Technologies and Computer Program Interfaces Which Correspond

From a managerial and technical point of view, the technology staffs supporting the design and release of a specific structural system are linked to each other and to the design project in a complex and highly interdependent manner. The various programs which make up a system of fully interfaced programs used to perform the technical analyses of a structural system must similarly be linked to each other and so organized internally and externally that data can be passed through the interfaces with minimum effort, and maximum visibility, control, and responsiveness.

Technology interfaces occur between Project, Master Dimensions, Weights, Static Air Loads, Dynamics, Flutter, Thermal, Aerodynamics and Performance, Stress, Fracture Mechanics, Fatigue, and Materials and Processes. Major quantities of information must be transferred between certain of these technology groups. These include geometry, section properties, weights data, air loads data, stiffness matrices, flexibility matrices, deflections, internal loads, and stresses.

4.2 ORGANIZATION REQUIRED TO IMPLEMENT MODELING ACTIVITY

The organization required to initiate a modeling activity focuses primarily on: 1) the scheduling of the support required, be it configuration, component development and test, layout review or drawing release, 2) the scheduling of the supporting finite element analysis major milestones, and 3) the commitment of computer facilities.

4.2.1 SST Drawing Release Schedules

The SST stress groups used the network of SAMECS and SAMECS peripheral programs to support layout review and drawing release of all of the body and wing structure. The goal in using finite element analyses was to obtain runs early enough in the design process for the results to have a solid impact on weight and fatigue lift. A minimum of three cycles of internal loads runs prior to 25% drawing release was the goal set for the mathematical model analysis group. The sequence of events by which release of structure was accomplished, focusing on the stress group activities primarily, was as follows, using the wing-body intersection structure as an example:

1) Obtain spar, rib, stiffener centerlines; upper and lower surface definition, frame, stringer, and cutout locations, construct additional finite element grid lines, and request from project loft Master Dimensions group the resulting node coordinates, offsets, and related geometry in such format that it is directly usable by the SAMECS system,

2) Obtain surface temperatures, structural element gradients throughout the substructure from the thermal group; external pressure distributions and associated load factors on punched cards from the loads group, unit and actual weights data from the weights group,
3) Add sufficient information to the data received in (2) to build and insure balance of the critical design load and nominal flight conditions of interest,
4) Calculate internal loads, deflections, stresses, and stiffnesses of structure using SAMECS,
5) Perform a preliminary sizing of the structure from the layouts using the results of this coarse grid finite element solution, and stiffness requirements,
6) Provide nominal flight condition deflections to project for preliminary deflected shape information to be used for jigging the vehicle,
7) Compare deflections and stiffness of coarse grid finite element models with those of dynamics, flutter, and loads groups to verify compatibility of stiffness representations arrived at using different fineness SAMECS models,
8) Supply reduced stiffness matrices to flutter, dynamics, and load groups for use in the next design data cycle in support of final drawing release,
9) Supply sizing of layouts to project for update of drawings, and to weights for update of weights data,
10) Refine the finite element grid to obtain better internal loads distributions in areas of high stress concentration and including the effects of post-buckling and diagonal tension directly in the finite element analysis. Use the SAMECS substructuring and freebody options to treat specific components, joints, etc. in greater detail as required.
11) Obtain new centerlines, node coordinates, and geometry corresponding to the refined finite element grid from the project loft Master Dimensions group,
12) Obtain updated thermal, weights, loads, and stiffness data from the respective staff and project groups,
13) Add sufficient information to the data received in (12) to build the load conditions of interest for final drawing release, jigging, and stiffness comparisons,
14) Calculate internal loads, deflections, stresses, and reactions of the structure using the refined finite element mathematical model,
15) Perform, in support of final drawing release, necessary stress, fatigue, safe life, fail safe, and stability analysis using the results of the refined model and recycles as required. Release drawings. Verify deflected shape for jigging with project, and stiffnesses with flutter, dynamics, and loads,
16) After final drawing release, initiate an additional pass through Master Dimensions, weights, loads, thermal, and flutter groups to obtain a final analysis for the structure as released. Steps taken are the same as (10) through (14).
17) Use the results of (16) to verify that all structure as released has positive or zero margins, to identify potential areas of weight saving, to verify stiffnesses used by flutter, dynamics and loads groups, to assist in location of strain gages and correlating with them in test, and to assist in definition of test loads.

4.2.2 Scheduling of Finite Element Analysis Major Milestones

All SST structural modeling activities of medium and large size were scheduled against a MMDMM (Major Milestones for Development of Mathematical Models) schedule modified to suit the requirements of each structural problem as necessary. The MMDMM schedule contains all of the major technical activities,

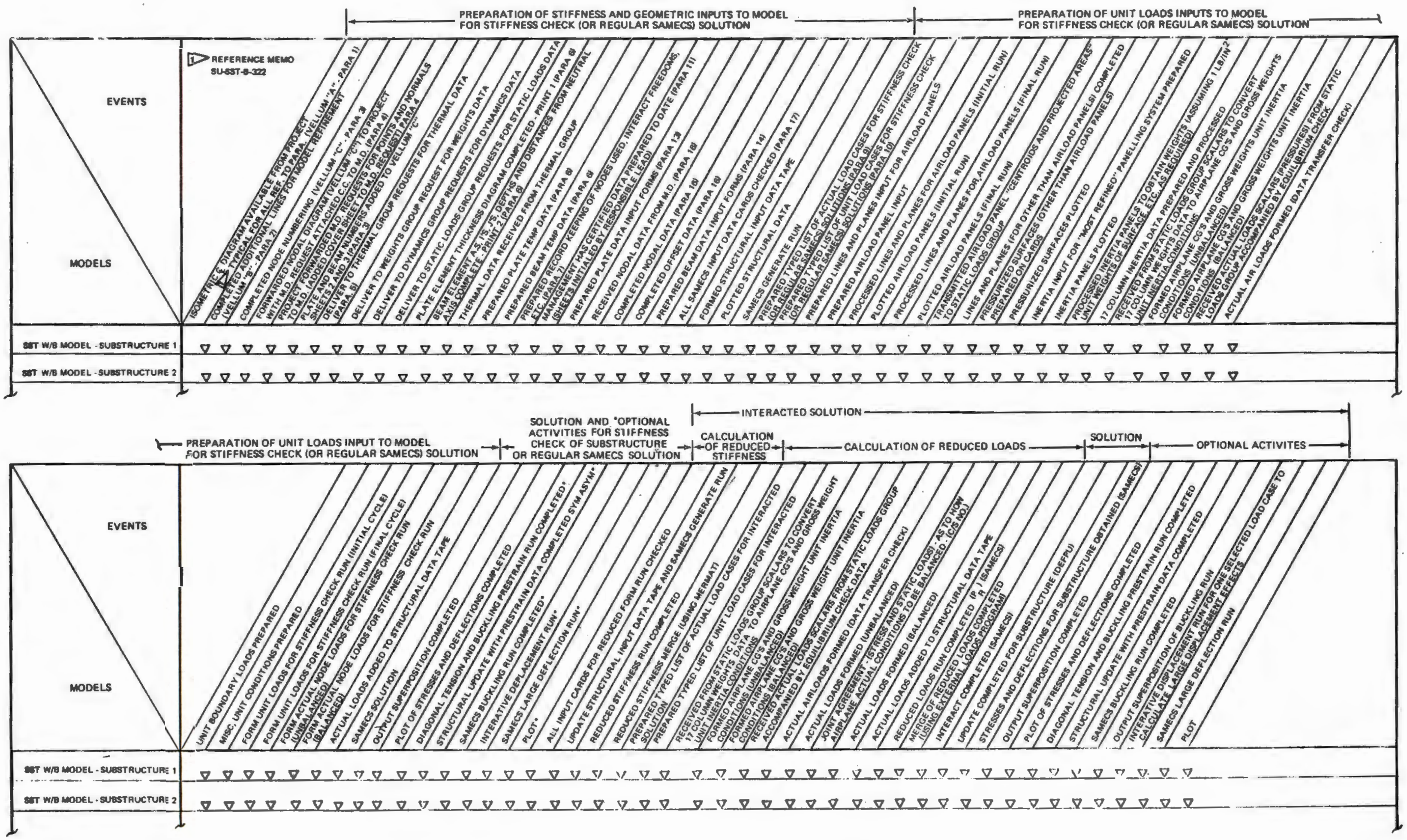

Figure 4.2.2: MAJOR MILESTONES FOR DEVELOPMENT OF MATH MODELS

arranged approximately in the order in which they most commonly occur, which must be accomplished in the development of finite element models. Each milestone in the MMDMM schedule is described in detail as part of the Boeing SAMECS technical procedures. The MMDMM schedule is shown in Figure 4.2.2. The schedule is subdivided into major phases which deal with: 1) preparation of the stiffness and geometric inputs to the model, 2) preparation of the unit loads inputs to the model, 3) preparation of the actual loads conditions for the stiffness check solution on substructures, 4) solution of stiffness check runs on substructures, 5) calculation of reduced stiffnesses, 6) calculation of reduced loads, 7) interact solution, and 8) post-processing activities including superposition, plotting, post-buckling, large displacement analysis, and iterations. There are 92 major milestones listed on the MMDMM schedule for each substructure including the interaction of that substructure.

4.2.3 Commitment of Technical Skills

The SST program retained an average of 6 engineers in finite element research and development for a six year period. Approximately 15 programmers performed finite element coding, consultation, and logistical support over the same time period. At the peak of the SST prototype drawing release phase Boeing employed 45 engineers who were experts in finite element methods and specifically familiar with SAMECS and the peripheral system of programs. This team of engineers produced multiple finite analysis solutions for over 30 different key substructures used to model the entire primary structure airframe. These structures averaged 800 nodes in complexity, and varied from 500 nodes to 2000 nodes. All of the SAMECS peripheral programs were used on these substructures depending upon the type of problems being dealt with. Most of the engineers involved with finite element modeling were also performing strength check and drawing release work. A complex analysis group within the stress group served as consultants in all facets of the analysis planning and execution. The average educational background of the 45 engineers involved was a Masters Degree, and many were equipped with their Doctorates. A definite management commitment to the use of finite element analysis procedures on the SST program was evidenced by the level at which technical skills were committed to the mathematical modeling activities. It was necessary that those used in modeling be well educated, trained, experienced, and familiar with the drawing release cycle in order to gain maximum utilization of the solutions results. On the SST program they were assigned in sufficient strength to allow the careful planning, preparation, checking and cross-checking required to produce quality results.

4.2.4 Commitment of Computer Facilities for Support of Analysis

The computing facility on which SAMECS and the SAMECS peripheral programs operated during the SST prototype layout review and drawing release was a back-to-back installation of two CDC 6600 computers, each with 131,000 core storage and 60 bit words. Three 6638 physical disks containing six logical disks were attached and switchable from computer to computer as required. The facility was equipped with high speed printers, card readers, tape drives, on-line card punch, on-line graphics display, teletypes, and highspeed remote terminals. Off-line data display capability included Cal Comp, Gerber, and SC 4020 plotters.

For approximately two years prior to the SST prototype cancellation the SST technology staffs required a dedicated CDC 6600 facility. Only because engineering management took the care to repeatedly forecast the required levels of computer support and present this picture to upper management was the staff adequately equipped and able to function without severe computing delays.

Without any question, the single most difficult task which faced engineering management was the procurement of adequate computer time, and in blocks of time sufficiently concentrated to correspond to and support periods of intensive technical effort.

4.3 QUALITY CONTROL OF THE MATHEMATICAL ANALYSES

Quality Control of the finite element analysis system is the key to the performance of the system. As larger problems are solved and the associated output data from these analyses increases it is necessary to carefully check all along the solution path to insure quality. Such control also provides a means for quickly detecting, locating, and resolving problems with a minimum expenditure of engineering and computer resources.

Many of the quality control checks in finite element analyses are exactly the same as would be used in any other analysis approach. The computer, however, opens up a large number of additional methods for exercising quality control. In most cases these new methods hinge upon the precision of the computer, which is an inherent by-product of the properly functioning machine. If engineers are precise in the way in which they feed information to the machine, then a lack of precision in the data returned indicates a malfunction or input error, and potential loss in accuracy. Though load conditions may be known to be somewhat inaccurate, they are always processed as balanced load conditions. The smoothness of node and offset geometry produced by a Master Dimensions program may be unnecessary for one load case and vital for another, unnecessary for one grid refinement and vital for a finer grid refinement of the same problem, particularly where the analysis being performed is using linear rather than large deflection analysis procedures. A very common cause of oscillating frame moments is the practice of scaling rather than the precise calculation of the coordinates.

Quality control of the analysis from generation of element sizing and mathematical geometry to final output display of results is necessary. In Sections 4.3.1 through 4.3.4 quality control brought about through the use of: 1) Master Dimensions, 2) equilibrium checks, 3) plotting, and 4) other means will be discussed.

4.3.1 Quality Control Through the Use of Master Dimensions

Procedures for precisely defining the geometry of the structural system through the use of the computer and a numerically controlled drafting machine were used on the SST to guarantee smooth and accurate geometry definition. The group which accomplished this activity in support of project is known as Master Dimensions. Master Dimensions also assists project with the production of drawings and provides manufacturing with jigging coordinates and magnetic tapes for operation of numerically controlled manufacturing equipment. It is possible to produce finite element analyses from the same geometry definitions that are used to produce the hardware, and proper geometric representation as well as smoothness of the structure geometry is assured. All Master Dimensions data is developed through project using project definitions for outside mold lines, centerlines, stringer and stiffener locations, etc. Any changes made by project in the lines are readily picked up by the math models. Quality control checks performed by Master Dimensions include multi-view plotting and comparisons with loft group equations and drawings.

4.3.2 Quality Control Through Equilibrium Checks

Following is the equilibrium check data that is automatically reported by the

SAMECS and SAMECS peripheral programs for use by the analyst in constructing finite element models:

1) unit air loads paneling projected areas and centroids,
2) unit weights paneling weights and c.g.'s,
3) airplane total weights and c.g.'s,
4) individual substructure total weights and c.g.'s,
5) air loads paneling equilibrium for each actual load case,
6) net inertia and air loads for each actual load case about the c.g. corresponding to each actual load case,
7) equilibrium of load cases output from loads transformation program versus equilibrium of load cases input to finite element program,
8) equilibrium of load cases input to finite element program against equilibrium of reactions output from finite element program,
9) force check equilibrium (force check involves applying a unit force or moment for every freedom in the mathematical model and then determining if every freedom load is precisely reacted by the reactions to eight significant digits),
10) thermal load case equilibrium is a computed zero.

4.3.3 Quality Control Through Plots of Mathematical Model Inputs and Outputs

Plots of the mathematical model are made throughout the analysis activity. Typical of the types of plots made are the following:

1) unit air loads panels
2) unit weights panels
3) node and element diagrams
4) element local coordinate systems
5) element connectivity
6) outside contours viewed from several angles
7) deflections of the mathematical model for load cases of interest to examine for soft spots in the idealization and reasonableness of deformations,
8) internal loads and stresses of the math model to determine if load paths are reasonable, to locate and understand stress concentrations, and to determine if the structure as sized is capable of carrying the loads with near zero margins.

4.3.4 Other Quality Control Checks

A uniform thermal expansion condition was processed as one of the load conditions for all SST mathematical models to verify that the elements are stress free (a computed zero). This condition gives the user an idea as to how accurate the internal loads are for the thermal conditions.

The conditioning number (number of significant digits lost in the solution for each freedom) is printed out for every freedom. This data is scanned and if more than four digits of the 14 available are lost, the idealization is scrutinized to determine the reason for the loss and the structure stiffness is corrected to reduce the digits lost to a maximum of four. This was a firm requirement for all interaction analyses.

All nodal diagrams, actual load conditions to be processed, initial reference data for stiffness and geometry definition, skin thicknesses, frame and stringer properties and other structural element properties are signed off by the responsible supervisors or their designees. All input forms and listings are checked. Boundary conditions and retained freedom data are double checked. (At no time is data checked by the same person who prepared the data). A management review of all models to verify that checks are complete, and to insure that management concurs with the structural idealization and loads is routinely held before data processing of a SAMECS run is initiated. Check sum reports are used to verify that appropriate tapes containing reduced stiffness matrices have been merged correctly. The criteria for superposing load conditions is reviewed and approved by management, as is the grouping of these load conditions for max-min purposes, and of course a check is made to see that the superposition was accomplished as intended.

Each mathematical model has its own peculiar check list selected from a list of over 100 categorical checks for each model or substructure. The wing-body joint analysis of the SST had 98 checks of the type described in Section 4.3 for each of the substructures.

4.4 TRANSFER OF THE OUTPUT FROM SAMECS TO THE TECHNOLOGY GROUPS

Several tools have been described earlier which are used for producing only significant output in a convenient form for use by the various engineering groups. They are the SAMECS program, the output superposition program, the output two-dimensional and three-dimensional plot program, the freebody program, and the margins of safety program. These programs are described in Section 3.0.

The various groups requiring information prepared cards requesting of the various programs the specific information and format required. Depending upon the priority of the request the results of such runs were obtained in from 2 hours to 24 hours on a normal backlog data processing day. This turn-around was in part realized because the SST program had available to it upon demand a computer fulltime. As an additional service to the engineers requiring data, the complex analysis subgroup frequently collected lists of requirements and produced the required results at the earliest possible time following completion of an analysis. The initial data produced by such runs was always processed, bound, and catalogued by that group.

4.5 SCOPE OF FINITE ELEMENT ANALYSES ON THE SST PROGRAM

The scope of the stress group mathematical modeling on the SST program was such that all of the primary structure and much of the secondary structure of that aircraft was treated using SAMECS. The mathematical model philosophy avoiding lumping of structural elements was adopted in virtually all of the body structure and in most areas of the empennage and wing structure. Figure 4.5 shows the major substructures treated on the SST and their approximate sizes. In most of the mathematical models the refinement of the grid was greater than that of the actual structure in order to obtain specific local stress concentration, and to include local large displacement effects. The total number of nodes used in the substructures of the SST airplane numbered approximately 30,000.

MAJOR SUBSTRUCTURE NUMBER	NODES	PLATES	BEAMS
1 FRONT SPAR	780	924	1560
2 FWD WHEEL WELL SPAR	808	822	1450
3 AFT WHEEL WELL SPAR	708	738	1314
4 REAR SPAR	802	808	1385
5 AFT TRANSITION	703	700	1300
6 WING, FORWARD	951	1420	1403
7 WING AFT.	1045	1836	1694
9 NOSE GEAR WHEEL WELL	1700	1700	3000
10 CARGO DOOR	1200	1200	2000
11 ENTRY DOOR	1080	1000	2080
12 FWD TRANSITION	945	940	1771
13 FOREBODY	1000	1800	2099
14 CAB	1286	1667	1577
16 AFT ENTRY DOOR AND PRESSURE BULKHEAD	900	800	1200
17 AFT BODY-FIN-AND RUDDER	1700	1800	1800
19 HORIZONTAL TAIL INCLUDING TORQUE BOX	1000	1280	1000
20 } CAB	800	800	100
21 } WINDOWS	900	900	100
24 }	700	580	200
25 } FOREBODY	808	900	200
26 } WINDOWS	900	800	200
27 }	700	600	200

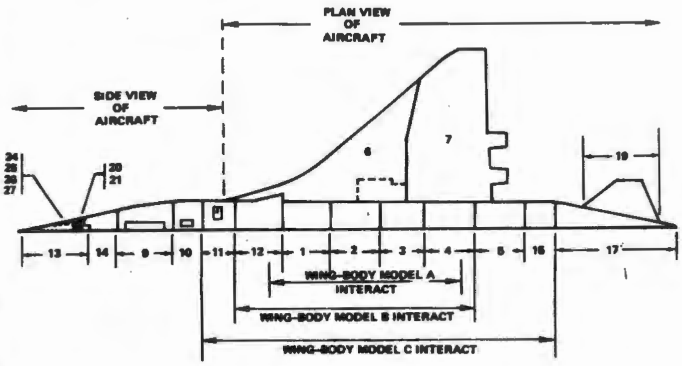

Figure 4.5: SCOPE AND SIZE OF TYPICAL SST SAMECS FINITE ELEMENT MODELS

4.6 COST EFFECTIVENESS OF USING THE FINITE ELEMENT METHODS FOR STRUCTURAL ANALYSIS OF THE SST

The question as to how cost effective the use of the computer is in obtaining internal loads and stress for aircraft hardware was clearly indicated in the course of the SST program. In many areas the benefits of finite element analyses were realized. These areas included the elimination or reduction of the scope of test programs (including quarter or half scale load distribution models and component testing that would have cost millions of dollars and added months of calendar time to schedules). The complex thermal environment, mechanical load and structural system defied effective treatment by any means other than the computer as mentioned in Section 2.0. Major emphasis was placed by management on the reduction of vehicle weight, and on the importance of structural reliability and life.

The question of cost effectiveness may be conservatively answered using only a portion of one of these realized benefits, namely that of weight reduction in the fuselage forward of the passenger compartment aft pressure bulkhead. Six key areas in which weight was saved are:

1)	Sections 43 and 45 frames	285 pounds per airplane
2)	Sections 46 and 47 frames	547 pounds per airplane
3)	Entry doors (6) and cargo door cutout	560 pounds per airplane
4)	Redistribution of load with wing in Section 46	150 pounds per airplane
5)	Redistribution of load with strake in Section 45 lower lobe.	100 pounds per airplane
6)	Reduction in Section 41 cab weight	300 pounds per airplane
		1942 pounds per airplane

Several other areas of weight saving in the body and all areas of weight savings in the empennage and wing are neglected. Boeing management determined that $150 to $200 per pound saved per airplane was a cost effective expenditure. Assuming a 200 airplane breakeven point in development and production costs, this meant that a maximum of $30,000 to $40,000 could be spent to remove a pound of recurring weight in the design of the SST. Based on these figures, and acceptable total cost of eliminating the 1942 pounds saved could have been as much as 58 to 77 million dollars.

The actual computing costs for all data processing by the SST stress group including the body, wing, and empennage sections was less than $1,250,000. If it were necessary to justify the entire stress group computer budget expended on the basis of the weight saved in the six key areas listed, the computing cost per pound of weight saved per airplane would be $3.21 which compares most favorably with the allowable $150 to $200 figures. The direct cost of computer time used by the stress group to save weight in the Sections 43, 45, 46, and 47 frames actually ranged from 15 cents to $2.75 per pound per airplane depending upon whether the frame in question was part of the major wing/body interaction or not.

The idea that the computer which consistently stays busy is cost effective is erroneous. Achievement of this goal also implies infinite turn-around for the unanticipated job. A cost effective use of the computer in solving structural problems requires that the computer system stand idle at times in order to accommodate the more active data processing periods of engineering schedules. In other words, engineering should be ready to pay a premium for guaranteed computing capacity and turn-around.

Computing facilities which are used by engineering to support drawing release schedules should be backed up by an equivalent facility to be used when machine breakdown occurs. It is not reasonable that a large engineering program on a strict schedule and costing hundreds of millions of dollars should count upon the proper functioning of a computer facility without a backup to support engineering and manufacturing schedules.

5.0 APPLICATION OF THE SAMECS FINITE ELEMENT ANALYSIS SYSTEM TO THE SST PROTOTYPE AIRCRAFT

Several mathematical models processed on the SST program serve to illustrate the application of finite element methods on the SST program, see Figure 4.5. Four models are presented: 1) the SST swing-wing wing-body joint analysis accomplished in 1969 before the switch to a delta wing, 2) the SST forward entry door model showing the level of detail treated around cutouts, 3) the SST crew compartment, and 4) the SST wing-body model B interact analysis completed in June of 1970.

5.1 THE SST SWING-WING/BODY JOINT ANALYSIS

A finite element model of the SST swing-wing/body joint was processed in 1967. Every stringer, frame, spar chord, rib chord, and stiffener was included in its offset location and with all section properties assigned. The entire analysis was accomplished within the confines of a single substructure, illustrating the capability of the SAMECS analysis system at the substructure level. The model contained 1899 nodes, 2191 plates, and 2978 beams in the half structure. Forty load conditions were processed including thermal and mechanical loads. The processing time for SAMECS in 1967 was 17 hours. The 1971 version of SAMECS would process the same problem in 7 hours. Figure 5.1 shows the model element diagram.

5.2 THE SST FORWARD ENTRY DOOR ANALYSIS

The forward entry door model illustrates the detail used around cutouts. This 1200 node, 1200 plate, 2000 beam substructure included every structural element discretely, and used still greater refinement in and around the corners of the entry cutout. Skin plate elements in the corners were 1¼" by 1¼" in size. Figures 5.2.1 and 5.2.2 show the model monocoque and local door cutout detail.

5.3 THE SST CREW COMPARTMENT ANALYSIS

The SST cab analysis included the body shell, window posts and sills, forward pressure bulkhead and floor. A portion of the node-element diagram is shown in Figure 5.3.1. Three-hundred unit-load and 80 actual load conditions were processed. The load cases covered critical design conditions including fatigue.

5.4 THE SST WING/BODY MODEL B INTERACT ANALYSIS

In June of 1970 the B model of the SST delta wing airplane was processed. The analysis involved 5939 nodes, 7484 plates, 10,566 beams, and approximately 2500 unit load conditions which were superposed to form 80 actual symmetric and anti-symmetric load cases. There were 2560 interact freedoms on the boundaries of the seven substructures. Loads included ascent, cruise, and descent thermal conditions, model quality check conditions, design and fatigue conditions, and a passenger compartment pressurization condition. Run time for this wing/body interact was 120 hours occupancy which included 50 hours central processing time. The SAMECS interact run processed for 3¼ hours occupancy. Thirty-thousand simultaneous algebraic equations were solved in the wing/body B analysis. Figure 5.4 is an element diagram for a typical substructure.

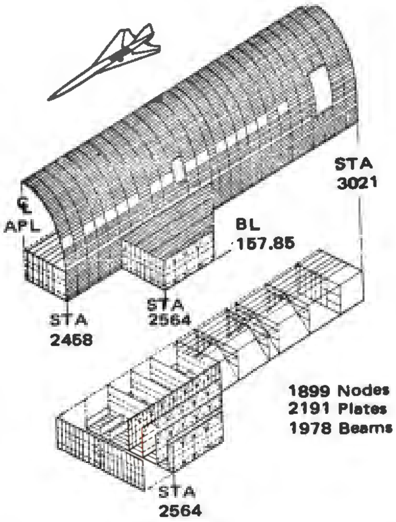

Figure 5.1: SST SWING-WING/BODY JOINT ELEMENT DIAGRAM

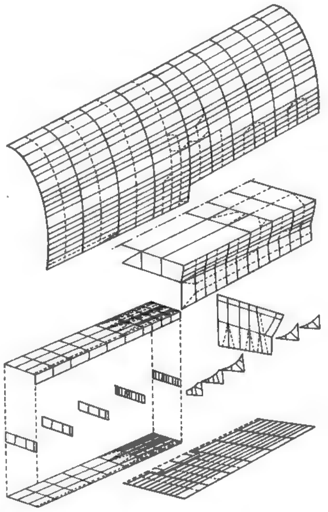

Figure 5.4: SST WING/BODY MODEL B SUBSTRUCTURE NO. 4 ELEMENT DIAGRAM

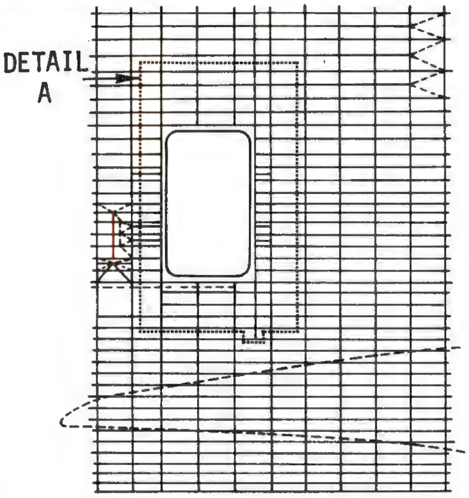

Figure 5.2.1: SST FORWARD ENTRY DOOR ELEMENT DIAGRAM

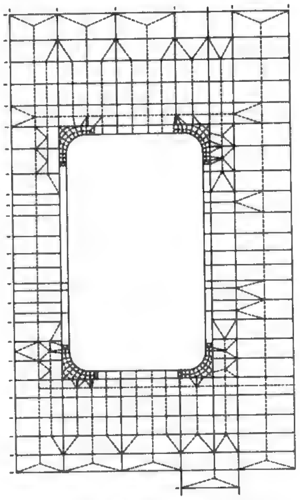

Figure 5.2.2: DETAIL A

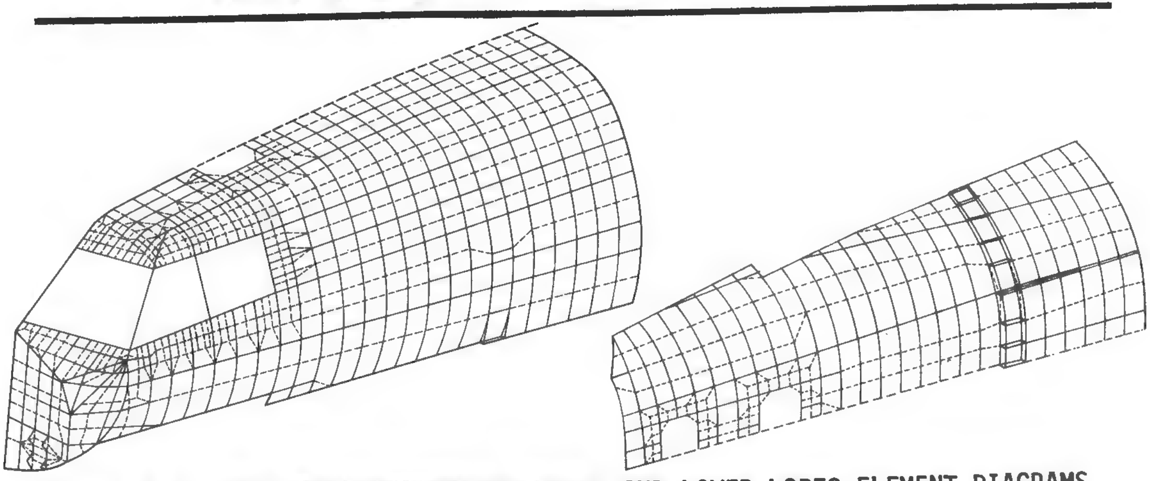

Figure 5.3: SST CREW COMPARTMENT UPPER AND LOWER LOBES ELEMENT DIAGRAMS

ACKNOWLEDGMENT

The author wishes to recognize Melvin A. Nelson and Andrew K. Hepler, members of SST Structures Staff Management, for their dedication to the improvement and extensive application of finite element programs and procedures for the Super Sonic Transport Program. Their commitment was indispensable to the existence of an internal loads analysis system of the quality described in this paper, and to the application of SAMECS to the SST and 747 aircraft.