

AGATE
(ADVANCED GENERAL AVIATION TRANSPORTATION EXPERIMENTS PROGRAM)



METHODOLOGY FOR SEAT DESIGN AND CERTIFICATION BY ANALYSIS

(REVISION A)

AGATE-WP3.4-034012-079-REPORT

SUBMITTED BY CESSNA AIRCRAFT COMPANY

AUGUST 17, 2001

Date of general release: August 31, 2001

Prepared for

Langley Research Center

National Aeronautics and Space Administration

Hampton, Virginia 23681-0001

REVISION

LETTER	DATE	ITEM	BY
N/C	05/28/01	Original release of report.	Terence Lim (Cessna Aircraft Company)
A	08/01/01	<ol style="list-style-type: none"> 1. Removed AGATE proprietary restriction statement from cover page. Document is released to the general public. 2. Revised Section 4.1.1.5. Head Injury Criteria (HIC) 3. Added section 4.5.5.1. Energy Balance 4. Moved and renumbered Section 3 Reference Publications to Section 2, and added references. 5. Section 2 Definitions was Section 3. 6. Revised Section 3.4 Stability of Codes. 7. Added Section 7 Acknowledgements 	Terence Lim (Cessna Aircraft Company)

TABLE OF CONTENTS

<u>SECTION</u>	<u>TITLE</u>	<u>PAGE</u>
1.	PURPOSE	1
2.	REFERENCE PUBLICATIONS	2
3.	DEFINITIONS	3
3.1	SEATING CONFIGURATION	3
3.2	SEATING SYSTEM	3
3.3	COMPUTER MODELING	3
3.4	STABILITY OF EXPLICIT CODES	4
4.	SEAT CERTIFICATION BY COMPUTER MODELING	6
4.1	GENERAL VALIDATION ACCEPTANCE CRITERIA	6
4.1.1	APPLICATION SPECIFIC VALIDATION CRITERIA	7
4.1.2	DISCREPANCIES	11
4.1.3	COMPUTER HARDWARE AND SOFTWARE	11
4.2	APPLICATION OF COMPUTER MODEL IN SUPPORT OF DYNAMIC TESTING	12
4.2.1	DETERMINATION OF WORST CASE FOR A SEAT DESIGN	12
4.2.2	DETERMINATION OF WORST CASE SCENARIO FOR SEAT INSTALLATION	13
4.2.3	DETERMINATION OF OCCUPANT STRIKE ENVELOPE	13
4.3	APPLICATION OF COMPUTER MODELING IN-LIEU OF DYNAMIC TEST	14
4.3.1	SEAT SYSTEM MODIFICATION	14
4.3.2	SEAT INSTALLATION MODIFICATION	14
4.4	SEAT CERTIFICATION PROCESS	14
4.4.2	CERTIFICATION PLAN	15
4.4.3	TECHNICAL MEETING	16
4.5	COMPLIANCE METHODOLOGY AND DATA REQUIREMENTS	17
4.5.1	PURPOSE OF COMPUTER MODEL	17
4.5.2	OVERVIEW OF SEATING SYSTEM	17

4.5.3	SOFTWARE AND HARDWARE OVERVIEW	18
4.5.4	DESCRIPTION OF COMPUTER MODEL	19
4.5.5	ANALYTICAL RESULT INTERPRETATION	21
4.5.6	MARGIN OF SAFETY	23
4.5.7	MINIMUM DOCUMENTATION REQUIREMENTS	23
4.5.8	RETENTION OF COMPUTER MODEL DATA DECK	23
5.	DYNAMIC SEAT COMPUTER MODELING GUIDELINE	24
5.1	UNITS	26
5.2	COORDINATE SYSTEM	28
5.3	OCCUPANT MODELS	29
5.3.1	ATB HYBRID II (PART 572 SUBPART B) OCCUPANT MODEL	30
5.3.2	MADYMO HYBRID II (PART 572 SUBPART B) DUMMY	34
5.4	MODELING STRUCTURAL ELEMENTS	40
5.4.1	METHOD 1 - MULTI-BODY TECHNIQUES	40
5.4.2	METHOD 2 - FINITE ELEMENT MODELING	42
5.4.3	METHOD 3 - HYBRID MODELING METHOD	54
5.4.4	MODELING FAILURE OF JOINTS OR FASTENERS	55
5.5	RESTRAINT MODELING	57
5.5.1	METHODS	57
5.6	MATERIAL MODELS	62
5.6.1	METALLIC MATERIAL MODELS	63
5.6.2	COMPOSITE MODELS	67
5.6.3	SEAT CUSHION FOAM MODELS	71
5.7	APPLYING BOUNDARY CONDITIONS	73
5.7.1	KINEMATIC CONSTRAINTS	74
5.7.2	CONTACT DEFINITION	76
5.8	LOAD APPLICATION	82

5.8.1	LOAD APPLICATION FOR 60 DEGREES PITCH TEST	82
5.8.2	LOAD APPLICATION FOR 10 DEGREES YAW TEST	84
5.9	FLOOR DEFORMATION	87
5.9.1	EXAMPLE FLOOR DEFORMATION SIMULATION USING MADYMO	87
5.9.2	EXAMPLE FLOOR DEFORMATION SIMULATION USING MSC/DYTRAN	88
6.	GENERAL DISCLAIMER	90
7.	ACKNOWLEDGEMENTS	91

FIGURE	LIST OF FIGURES	PAGE
Figure 5-1	Computer Modeling in Seat Design	24
Figure 5-2	Example Unit Specification	27
Figure 5-3	Model Coordinate System Orientation	29
Figure 5-4	ATB HII Occupant Model	32
Figure 5-5	Finite Element MSC/DYTRAN ATB Model	34
Figure 5-6	MADYMO HYBRID II (PART 572 Subpart B) DUMMY	35
Figure 5-7	Multi-body model	42
Figure 5-8	FE Modeling Flowchart	44
Figure 5-9	Spring Element	46
Figure 5-10	Rod Element	47
Figure 5-11	Beam Element	48
Figure 5-12	Shell Element	49
Figure 5-13	Solid Element	50
Figure 5-14	MSC/DYTRAN FE Model	51
Figure 5-15	Exploded View of FE Seat	53
Figure 5-16	MADYMO Hybrid Modeling Model	54
Figure 5-17	MADYMO 4-Point Restraint Before Pre-simulation	60
Figure 5-18	MADYMO Hybrid Belt After Pre-simulation	61
Figure 5-19	Elasto-Plastic Material Model	64
Figure 5-20	Example MSC/DYTRAN Input for Strain Rate Material	66
Figure 5-21	Example LS-DYNA3D Input for Strain Rate Material	66
Figure 5-22	Example MADYMO Input for Strain Rate Material	66
Figure 5-23	User Defined Shell Integration Points	68
Figure 5-24	Foam Impact Test	72
Figure 5-25	Stress-%Crush Foam Data	73

Figure 5-26 Example of FOAM1 material model	73
Figure 5-27 Rigid Connections	75
Figure 5-28 Example RCONN Input Deck	76
Figure 5-29 Contact Applications	78
Figure 5-30 MSC/DYTRAN Surface Contact Definition	79
Figure 5-31 MADYMO Multi-Body Contact	81
Figure 5-32 Multi-body Contact Definition	81
Figure 5-33 ATB 1 G Load Application Pitch Test	83
Figure 5-34 MSC/DYTRAN Load Application Pitch Test	84
Figure 5-35 Test 1 Applied Loads	84
Figure 5-36 Test 2 Applied Loads	85
Figure 5-37 MSC/DYTRAN Load Application Yaw Test	86
Figure 5-38 Floor Deformation Using MADYMO	88
Figure 5-39 Floor Deformation Using MSC/DYTRAN	89
Figure 5-40 MSC/DYTRAN Pitch and Roll Simulation	90

FOREWORD

A methodology for demonstrating compliance with FAR Part 23.562 by computer modeling analysis technique, validated by dynamic seat tests, was generated by the AGATE Advanced Crashworthiness Group. This task was performed in collaboration with the Federal Aviation Administration Small Airplane Directorate. Nothing in this document shall supersede applicable laws and regulations. This document was developed by The Cessna Aircraft Company under the AGATE crashworthiness program, and is approved by the principal members of the Integrated Design and Manufacturing Technical Council for public release under the terms of the Joint Sponsorship Research Agreement. This document may be reproduced and distributed without restrictions. Any improvements, beneficial comments or clarification needed regarding the contents of this document shall be forwarded to:

Advanced Crashworthiness Group
Intergrated Design and Manufactring Technical Council
c/o AGATE Alliance Association Inc.
3217 N. Armistead, Ste. M
Hampton, VA 23666-1379

1. PURPOSE

The purpose of this document is to provide guidance for demonstrating compliance with FAR Part 23.562 by means of computer modeling analysis techniques. It defines the acceptable applications, limitations, validation processes and minimum documentation requirements that are involved when substantiation by computer modeling is used to support a seat certification program.

This document also provides guidance and lists specific examples on the methodologies associated with generating occupant crash simulation. The intent of this document is to provide an engineer with background in transient finite element modeling with sufficient details to develop a seat/occupant computer model that may be successfully employed for design and certification. Since the practice of computer modeling is highly dependent on the state of hardware and software at the time of the release of this document, future enhancement may effect portions of the guideline, and appropriate update to this document will be required.

It is recognized that there may be more than one possible approach in generating a seat/occupant computer model. Therefore, the methodologies and examples presented in this document should not be construed as the only method of performing a computer analysis of the seat/occupant system. Other modeling techniques, subjected to reasonable validation, may be acceptable and should be coordinated with the FAA if the data is to be used for certification purposes.

2. REFERENCE PUBLICATIONS

- Code of Federal Regulations, Title 14 Part 21 Certification Procedures for Products and Parts.
- Code of Federal Regulations, Title 14 Part 23 Airworthiness Standards: Normal, Utility, and Acrobatic Category Airplanes
- U.S. Department of Transportation FAA Order 8110.4A Type Certificate Process
- Advisory Circular 23.562-1 "Dynamic Testing of Part 23 Airplane Seat Restraint/Systems and Occupant Protection", 1989.
- SAE 8049 Rev.A "Performance Standard for Seats in Civil Rotorcraft, Transport Aircraft, and General Aviation Aircraft", 1997.
- SAE J211 Instrumentation for Impact Test, SAE Recommended Practice, March 1995.
- *Articulated Total Body Version V.1 User's Manual*, United States Air Force Research Laboratory 1998.
- *MSC/DYTRAN User's Manual Version 4.7*, The Mac-Neal Schwendler Corporation 1999.
- *MADYMO User's Manual 3D Version 5.4*, TNO-MADYMO 1999.
- *MADYMO Database Manual 3D Version 5.4*, TNO-MADYMO 1999.
- *MADYMO Theory Manual 3D Version 5.4*, TNO-MADYMO 1999.
- *LS-DYNA Theoretical Manual*, Livermore Software Technology Corporation 1998.
- *LS-DYNA User's Manual Version 940*, Livermore Software Technology Corporation 1997.
- *Finite Element Procedures in Engineering Analysis*, K.J. Bathe 1982.
- *Solutions Method*, T.Belytschko, W.K.Liu and B. Moran 1999.

3. DEFINITIONS

3.1 SEATING CONFIGURATION

The aircraft interior floor plan, which defines the seating positions available to passengers during take-off, landing and in-flight conditions.

3.2 SEATING SYSTEM

A seating system is comprised of the seat structure, upholstery and restraint system.

3.3 COMPUTER MODELING

The use of computer based finite element or multi-body transient analysis to simulate the physical crash event. These codes typically follow an explicit formulation. The following combination of computer codes and occupant models have been tested for use in the design and certification of dynamic seats.

1. MADYMO^{®1} transient finite element/multi-body software and the MADYMO[®] 50% Part 572 Subpart B (Hybrid II) occupant model.
2. MSC/DYTRAN^{®2} transient finite element software and the ATB^{®3} (Hybrid II) occupant model
3. LS-DYNA3D^{®4} transient finite element software and the MADYMO[®] 50%

1 [®] MADYMO is a registered trademark of TNO Road-Vehicles Research Institute

2 [®] MSC/DYTRAN is a registered trademark of the MacNeal-Schwendler Corporation

3 [®] ATB is a public domain code developed and maintained by Wright Patterson Air Force Base

4 [®] LS-DYNA3D is a registered trademark of the Livermore Software Technology Corporation

Part 572 Subpart B (Hybrid II) occupant model.

3.4 STABILITY OF EXPLICIT CODES

Most transient explicit finite element codes employ direct integration methods, and take advantage of the numerical effectiveness of integration schemes such as the central difference methods, Wilson- θ or Newmark β -methods. These integration schemes attempt to satisfy equilibrium only at discrete time intervals (Δt) rather than for the duration of the analysis.

The accuracy and stability of the solution is highly path dependent, and relies heavily on the interpolated values of displacements, velocities and accelerations within each time step interval. The inherent numerical instabilities encountered with explicit dynamic analysis codes are discussed in detail, most notably by Bathe and Belytschko in their respective publications (reference Section 2). *The solutions are therefore conditionally stable*, a trade-off for the simplicity and cost effectiveness of the methods. The stability of the explicit methods is a function of the critical time step Δt_{cr} defined as

$$\Delta t_{cr} = \min \frac{l_e}{c}$$

where l_e is the effective length of the smallest element, and c is the wave speed (a function of material stiffness). In other words, the time step selected for the analysis must be smaller than the time for

the stress wave to cross the smallest element in the finite element mesh. Otherwise, the solution can grow without bound and deviate from stability, and thereby, producing erroneous results.

In theory, the most accurate solution is obtained when an integrating time step equivalent to the stability limit is chosen. Commercial codes, such as MADYMO or LS-DNA3D, attempt to offset the problems of numerical instability by automatically regulating and constantly updating the time interval used throughout the analysis. Although the user may choose an initial time step to begin the analysis, the program will calculate the critical integration time step, and will either terminate or default to the critical time step if the user input time step is larger than the minimum.

4. SEAT CERTIFICATION BY COMPUTER MODELING

Computer analysis may be used to substantiate a seat system design that is subjected to the certification requirements of FAR Part 23.562 after it has been correlated to the validation acceptance criteria specified in Section 4.1. The validation must be performed on a baseline seat design that has demonstrated compliance, by test, to 14 CFR 23.562.

Once validated, the model may then be utilized for certification purposes under the conditions specified in Section 4.2 and 4.3. Further utilization of computer analysis for demonstrating compliance beyond the conditions specified in Sections 4.2 and 4.3 will occur as the experience base of industry grows.

4.1 GENERAL VALIDATION ACCEPTANCE CRITERIA

The model is considered validated and may be used as means of demonstrating compliance if the validation acceptance criteria specified in this section have been demonstrated. The criteria will allow for some subjective interpretation as long as the basis of such interpretation is consistent with good engineering judgment. Such interpretation shall also be commensurate with the basis of the regulation, and the level of correlation required of the applicant shall not be imposed to tolerances beyond that observed in a dynamic test. The validation acceptance criteria are as follows:

1. The model must be reasonably validated against a dynamic test.

2. The model can be utilized for substantiation under similar conditions that the model was validated against.
3. The general pre-impact occupant trajectory, verified by visual comparisons, should correlate against test data.

In addition to the general validation criteria above, the model has to correlate to the following application specific criteria defined in Section 4.1.1.

4.1.1 APPLICATION SPECIFIC VALIDATION CRITERIA

The intent is to have the applicant validate -in addition to the general validation criteria- parameters that are relevant to the application of the model. This will remove undue burden from the applicant to perform validation for other parameters that may not be used in the certification. The relevant application specific validation criteria should be established and agreed by the FAA ACO, and listed in the certification plan. Test data used to validate the model should be included as an appendix in the analysis report. The computer model is considered validated if reasonable agreement between analysis and test data can be shown. Acceptable correlation methods related to each application specific validation criteria are defined in Section 4.1.1.1 to 4.1.1.6.

4.1.1.1 OCCUPANT TRAJECTORY

Occupant trajectory describes the overall motion of the occupant. The trajectory of the occupant (such as headpath) determined by analysis

may be compared to high-speed video obtained from dynamic tests. Validation may be established by visual comparison or by over-laying space (xy, yz or zx) time-history plots obtained from the analysis to calibrated photometric data obtained from dynamic tests.

4.1.1.2 STRUCTURAL RESPONSE

The computer model, used for structural certification, may be validated by correlating the following structural performance criteria to dynamic test.

4.1.1.2.1 INTERNAL LOADS

Internal loads such as floor reaction loads are a required means to show correlation. Reasonable agreement between the peak resultant floor reaction load obtained in the analysis and test data should not exceed 10%.

4.1.1.2.2 STRUCTURAL DEFORMATION

Reasonable agreement should be obtained between the mode of structural deformation obtained by analysis and test data for members that are critical to the overall performance or structural integrity of the seat or seating system. Validation may be established by visual comparisons or by over-laying space (xy, yz or zx) plots obtained from the analysis to photometric data obtained from dynamic tests.

4.1.1.3 RESTRAINT SYSTEM

Compliance with shoulder harness load is defined in FAR Part 23.562(c)(6). Validation of the restraint system may be obtained by correlating the analysis belt load force-time history to test data. The phase and maximum value force-time history profile should correlate within 10% of dynamic test data. This would ensure that in the analysis, the energy from the occupant as a result from inertia forces are transferred appropriately to the seat and vice versa. Additional parameters such as belt pay-out or permanent elongation may be correlated if similar measurements were recorded during dynamic test.

4.1.1.4 INJURY CRITERIA

Validation of the injury criteria may be obtained by correlating the analysis time history plots to test data. In general, the level of deviation in the injury criteria between analysis and test data should not exceed 10%.

4.1.1.5 HEAD INJURY CRITERIA (HIC)

Compliance with Head Injury Criteria is defined in FAR Part 23.562(c)(5). The regulation specifies HIC to be calculated during the duration of the major head impact, and the maximum allowable HIC limit is 1,000 units. The selected time interval¹ used in calculating HIC may not exceed 50 milliseconds. If the HIC evaluation involves head impact with airbags, FAA will determine the appropriate HIC limit and time interval criteria². In either case, the time interval used to

evaluate HIC in the analysis should be selected to match the time interval size used to evaluate HIC in the test. Because HIC is a maximizing function, the reported time duration³ that produces the maximum HIC need not match. The analysis is validated for HIC if the following correlations between analysis and test data are established.

1. The phase and profile of the acceleration time-history plot for resultant head accelerations.
2. The average resultant "G" loading as measured from the center of the head center of gravity.
3. The HIC calculation, using the same time interval.

¹ The term 'time interval' used in this section is defined as the duration between the initial and end time which the user selects to calculate HIC, which should correspond to the duration when the ATD is exposed to head impact on airplane interior features.

² Shorter HIC evaluation time intervals and lower HIC limits are used in the automotive regulations (46 CFR 571.208) to account for head/airbag interactions, and may be appropriate in some airplane certification. The validation of computer models using a HIC limit other than that specified in 14 CFR 23.562 should be approved by the FAA.

³ The term 'reported time duration' used in this section is defined as two points in time in the head acceleration profile that produces the maximum HIC. This reported time duration is not user defined, and is based on the outcome of the HIC algorithm.

4.1.1.6 SPINE LOAD

Compliance with spine load is defined in FAR Part 23.562(c)(7). The maximum allowable limit is 1,500 pounds. The phase and maximum value force-time history profile for spine load obtained in the analysis should be correlated to the dynamic test.

4.1.2 DISCREPANCIES

Failure to satisfy all validation criteria does not automatically preclude the model from being validated. The applicant and the FAA ACO engineer should evaluate if the deviations will have a detrimental impact on the model to sufficiently predict the crash scenario, and to determine if deviations from the validation criteria are acceptable. In addition, the applicant may present evidence to show that the deviation is within the inherent reliability and statistical accuracy of the test results. Discrepancies between results obtained from analysis and test data should be quantified.

4.1.3 COMPUTER HARDWARE AND SOFTWARE

The model should be used for certification on the same hardware and software platform that the validation was conducted. The model should be developed using the production version of the software. Beta releases are not allowed. If the computer model is transferred for use on a different platform, the applicant must re-validate the model as necessary to ensure that the results do not reflect any significant differences.

4.2 APPLICATION OF COMPUTER MODEL IN SUPPORT OF DYNAMIC TESTING

The purpose of this section is to encourage the use of analysis to reduce the number of full-scale dynamic test that are required to certify a seat design or installation. This is beneficial in certifying seats that are based on the same design concept, but may differ structurally to accommodate a particular installation. A final certification test is normally required to certify the worst-case seat design or installation.

When the intent of the computer model is to provide engineering analysis and rationale in support of dynamic testing, the results from the computer model may be used for, but are not limited to, the following conditions specified in Section 4.2.1 through 4.2.3. Additional conditions, which are currently not defined, shall be coordinated with the local FAA ACO and approved in the certification plan.

4.2.1 DETERMINATION OF WORST CASE FOR A SEAT DESIGN

Upon completion of the computer analysis, the results from the simulation may be used to determine the worst case or critical loading scenario for a particular seating system. This includes

1. Identifying components of seat structures that are critically loaded.
2. Selection of critical seat tracking positions.

3. Determine the direction of floor deformation to produce worst case loading on seat frame.
4. Evaluation of restraint system.
5. Selection of worst-case seat cushion build-up.
6. Evaluation direction of yaw condition to address loading on seat frame and movement of occupant out of restraint system.

4.2.2 DETERMINATION OF WORST CASE SCENARIO FOR SEAT INSTALLATION

For seats, which have been shown by analysis or test to be similar, computer analysis may be used to select the worst case seating system in the seating configuration for dynamic testing. Each seating system shall be analyzed in its production installation configuration.

Examples where analysis may be used to determine a worst case seating system may include:

1. Seating system installed in an over-spar versus a non-over spar configuration.
2. Seating system installed at different positions in the fuselage, which results in varying restraint anchor positions relative to the occupant and seat structure.

4.2.3 DETERMINATION OF OCCUPANT STRIKE ENVELOPE

The results of the computer analysis may be used to determine the occupant strike envelope with aircraft interior components. Each seating system shall be analyzed in its production installation

configuration. The occupant strike envelope can then be used to determine if a potential for head strike exist, and if so, which items are required in the test setup during the HIC evaluation tests.

4.3 APPLICATION OF COMPUTER MODELING IN-LIEU OF DYNAMIC TEST

The purpose of this section is to encourage the use of analysis to eliminate dynamic testing on certified seats. When the intent of the computer model is to provide engineering data in-lieu of dynamic testing, the results from the computer model may be applied to the following conditions:

4.3.1 SEAT SYSTEM MODIFICATION

Analysis based on computer simulation may be used to re-substantiate seat designs which have been modified from the TSO'd or certified configuration. No additional testing is required.

4.3.2 SEAT INSTALLATION MODIFICATION

Analysis based on computer simulation may be used to re-substantiate seat installations. The primary application is to show compliance for HIC and occupant body-to-body contact as a result of changes in seat arrangements.

4.4 SEAT CERTIFICATION PROCESS

This section contains certification guidelines when computer modeling is utilized as supporting engineering data to demonstrate compliance

with FAR Part 23.562. It defines the procedures that are involved with regards to FAA coordination, guidelines for the preparation and validation of the computer model, and the minimum documentation requirements for FAA data submittal.

4.4.1.1 FAA COORDINATION

The FAA coordination process used in this document has been extracted from FAA Order 8110.4A. FAA coordination is essential in ensuring the proper and timely execution of any certification program. Specific guidelines are presented to assist in the implementation of computer modeling as a means of compliance.

4.4.2 CERTIFICATION PLAN

The use of computer modeling as technical data to support the establishment of dynamic test conditions or in-lieu of dynamic test shall be negotiated with the FAA during the preliminary and interim Type Certification Board (TCB) meeting. The applicant's role is to:

1. Acquaint the FAA personnel with the project
2. Discuss and familiarize the FAA with the details of the design
3. Identify, with the FAA, applicable certification compliance paragraphs.
4. Negotiate with the FAA where the applicant will utilize computer modeling, specify its intent and purpose for the analysis.

5. Establish means of compliance, either by test, computer modeling or both with respect to the certification requirements.
6. Establish the validation criteria for the computer model relative to its application for certification.
7. Prepare and obtain FAA ACO approval of the certification plan.

4.4.3 TECHNICAL MEETING

The details of the computer model are defined during scheduled technical meetings held with the FAA ACO. The applicant should prepare a document for the FAA describing the purpose of the analysis, the validation methods and data submittal format. As a minimum, the following items should be contained in the document:

1. Description of the seat system to be modeled.
2. Selection of software for the analysis.
3. A description of how compliance will be shown.
4. Validation method.
5. Interpretation of results.
6. Substantiation documentation and data submittal package.

The document, hereby referred to as the analysis report, should be developed in conjunction with the seat design evaluation phase, and approved by the FAA as early in the certification program as possible.

4.5 COMPLIANCE METHODOLOGY AND DATA REQUIREMENTS

The following sections define the methodology for showing compliance and minimum documentation requirements when computer modeling is submitted as engineering data. As a minimum, the analysis report should contain the following:

4.5.1 PURPOSE OF COMPUTER MODEL

The applicant should define the purpose of the computer model and a list of the FAR requirements relevant to the certification of the seating system. Emphasis should be given to describe how the computer model would be used to demonstrate compliance for each stated requirement.

4.5.2 OVERVIEW OF SEATING SYSTEM

Provide an overview of the design of the seating system. Describe the seat layout in the aircraft, restraint type, and attachment to the airframe. If applicable, state the adjustment positions required during take off and landing. Discuss special occupant protection features included in the design.

4.5.2.1 SEAT STRUCTURE

Describe the critical components of the seat, the primary load paths and energy absorbing features. Provide a description on how the seat(s) are attached to the airframe. List the material properties of

the primary structural and energy absorbing components, and specify the method of fabrication.

4.5.2.2 RESTRAINT SYSTEM

Provide a description of the restraint system and any other devices that are intended to restrain the occupant in the seat or reduce the occupant's flail envelope under emergency landing conditions. This may include the shoulder and lap belts, load limiting devices, belt locking devices and pretensioners. Describe how the restraint system and its devices are attached or secured in position.

4.5.2.3 UNIQUE ENERGY ABSORBING FEATURES

Unique energy absorbing features are components, other than the seat and restraint system, that are designed to limit the load into the seating system or occupant. Examples include energy absorbing sub-floor structure and inflatables that are not mounted on the seat.

4.5.3 SOFTWARE AND HARDWARE OVERVIEW

The analysis report should contain a brief description of the software and hardware used to perform the analysis, and should include the following information:

1. Type and platform of computer hardware
2. Software type and versions
3. Basic software formulation.

4.5.4 DESCRIPTION OF COMPUTER MODEL

The analysis report should contain a detailed description of the computer model. This includes providing rational to the following:

4.5.4.1 ENGINEERING ASSUMPTIONS

Assumptions that are made in the analysis should be documented.

Assumptions may include simplification of a physical structure, the use of a particular material model, methods used for applying boundary conditions, method of load application, etc. Discuss the validity of the assumptions and provide rational support for the assumptions. If required, demonstrate that the assumptions do not negatively affect the results.

Components that are not critical to the performance of the seating system and do not influence the outcome of the analysis may be omitted from the model. A list of all components that are excluded from the analysis shall be documented. Comments should be included to justify its exclusion.

4.5.4.2 DISCRETIZATION OF PHYSICAL STRUCTURE

A description of the finite element mesh of the structure should be provided in the analysis report. It should describe how the critical components of the structure were modeled and provide the rational for the selection of element types that were used to represent the structure.

4.5.4.3 MATERIAL MODELS

Data of material models in the analysis should be documented in the analysis report. List the materials used by the analysis software and provide a general description. Document the source of material data.

Material data acquired through in-house tests must be supported by appropriate documentation that describes the basis of such test, test methods, and results. This includes proprietary data.

4.5.4.4 CONSTRAINTS

Constraints are boundary conditions applied in the model. This includes single and multi-point constraints, contact surfaces, rigid walls and tied connections. Document the boundary conditions applied in the model. Discuss how the model boundary conditions correspond to the test conditions. Provide a description on all contact definitions and nodal constraints.

Document the values used to represent frictional constants and the validity of such values.

4.5.4.5 LOAD APPLICATION

Loads that are applied in the computer model include concentrated forces and moments, pressure, enforced motion and initial conditions. Describe how external loads are applied to the model. List the source of the crash pulse and include a copy of the profile in the appendix.

4.5.4.6 OCCUPANT SIMULATION

The use of appropriate occupant models is dependent on the objective of the analysis. The use of the appropriate occupant model should be negotiated with the FAA. If the analysis is used to certify to the requirements of FAR 23.562(b)(1) and (b)(2) conditions, then a validated occupant model representing a 50th percentile male per 49 CFR Part 572 Subpart B or equivalent approved dummy should be used. Descriptions should be included in the analysis report on the development and validation of the occupant model.

4.5.4.7 GENERAL ANALYSIS CONTROL PARAMETERS

General analysis control parameters are features of a program that control, accelerate and terminate the analysis. It may also include parameters that enhance the performance of the software for the purpose of reducing the computational time, and subroutines that are employed to facilitate post-processing of results.

A summary of the control parameters used for a particular analysis should be documented. Parameters that may influence the outcome of the analysis should be justified. For example, the analyst should show that artificial scaling of mass for the purpose of reducing computational time is acceptable and does not negatively influence the results of the model.

4.5.5 ANALYTICAL RESULT INTERPRETATION

This section contains guidance and recommendations for the output, filtering and the general methods of reporting analytical data. The purpose is to achieve uniformity in the practice of reporting analytical results. The use of the following recommendations will provide a basis for meaningful comparison to test results from different sources.

4.5.5.1 ENERGY BALANCE

A summary of the ratio of initial energy to final energy, and a comparison of hourglass energy to total energy should be provided. The hourglass energy should not exceed 15% of total energy. In addition, the deformation modes associated with the presence of hourglass energy should be evaluated to determine if they are located at critical components of the structure, upon which, and an assessment of the hourglass modes and its influence on the accuracy of the analysis be determined. The model should be corrected as required if the appropriate energy balance is not attained.

4.5.5.2 DATA OUTPUT

Data from transient analysis should be generated at channel class 1000. The purpose is to maintain an equivalent practice with the instrumentation requirement specified in SAE J211 so that a meaningful comparison to test data may be performed.

If the output of the data channels is dependent on the integration time step of the analysis, and its sample rate is higher than channel

class 1000, the data should be reduced to be consistent with channel class 1000 prior to filtering. A deviation should be documented in the analysis report.

4.5.5.3 DATA FILTERING

The filtering practices of SAE J211 shall apply for all applications (reference in Section 5 of the SAE J211 document for the recommended channel class filtering).

4.5.6 MARGIN OF SAFETY

Margin of safety applies only to structural substantiation and should show a positive margin of safety. Injury pass/fail criteria shall not exceed the maximum value as specified in 14 CFR 23.562(c).

4.5.7 MINIMUM DOCUMENTATION REQUIREMENTS

The FAA data submittal package to show compliance with FAR 23.562 by means of computer modeling should contain the following:

1. Report of the analysis.
2. Video of the computer model simulation.

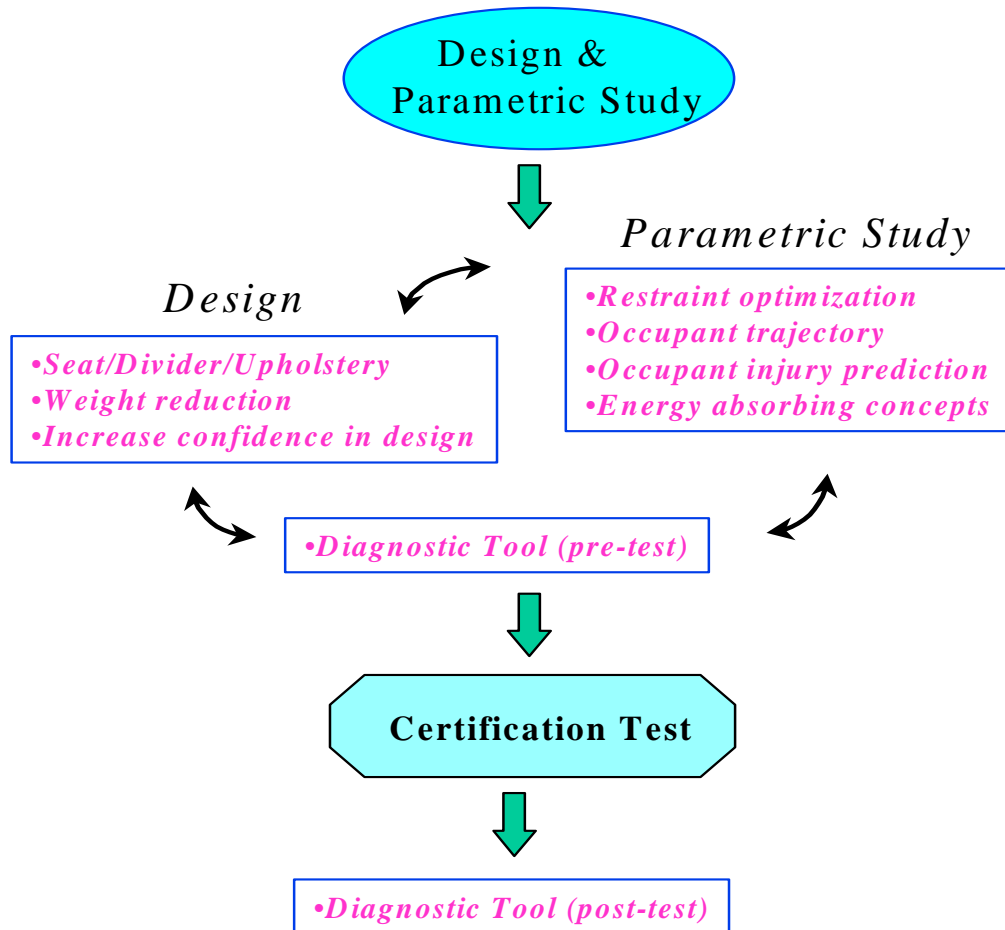
4.5.8 RETENTION OF COMPUTER MODEL DATA DECK

A copy of the computer model data deck used for substantiation should be archived for reference purposes. The archived copy of the data deck should include the date and the final revision number of the model.

5. DYNAMIC SEAT COMPUTER MODELING GUIDELINE

This section presents some of the methods used to develop a computer model of the occupant and seating system. The examples presented reflect the versions of the software used at the time of the release of this document (reference Section 3.0). When used effectively, computer models can reduce the cost and certification schedule significantly. Figure 5-1 shows a flowchart on the use of computer modeling in the dynamic seat design process.

Figure 5-1 Computer Modeling in Seat Design



In the preliminary design phase, computer modeling is used to perform numerous parametric studies to investigate different energy absorbing concepts and establish design parameters to meet the structural and occupant loads. Simple restraint models are generated to predict occupant trajectory and determine, optimize restraint design and determine the approximate anchor mount positions. Information from the parametric analysis is used to produce the prototype seat design. The prototype seat is then evaluated for fit and function, and modifications made to refine the design.

More details are added to the computer models as the seat design moves from the prototype to the first production concept design. The analysis is performed to obtain an accurate prediction of structural and occupant response, and in particular, occupant loads with respect to the dynamic pass/fail criteria. The objective is to reduce the risk of failure and the need to re-test during the certification program. In this phase, detailed finite element models are used to generate cross-section properties of beam structures that can withstand the dynamic load.

Iterations in analysis are performed to obtain an optimal stiffness-to-weight ratio. Interior components such as glareshield, instrument panels and side-ledges are added to the model to predict the head injury criteria. Seat cushions, seat pans or energy absorbing devices are modeled to predict spine load. Floor deformation analysis is performed to determine if the seat structure is able to react the induced pre-stress and crash load without failure. The simple axial

belt model used in the parametric analysis is replaced with 2-D finite element belt model to provide better occupant trajectory predictions.

An evaluation test is conducted on the seat design and appropriate changes are made based on the test results. The design and analysis cycle is iterated until a satisfactory design is attained, and the seat program proceeds to the certification phase.

Computer models can also be utilized as a post-test diagnostic tool. Well-prepared models can sometimes help identify anomalies that occurred during a test that are linked to bad instrumentation channels. The computer model helps establish the range or approximate values that a measuring device may produce, such as shoulder harness load or head acceleration. The output from the computer model can be compared with actual test signals to determine if the test data are physically possible or if the signals are compromised by noise or faulty instrumentation.

5.1 UNITS

Transient finite element modeling requires the use of a consistent set of engineering units for the fundamental measures of length (L), time (T), mass (M) and derivative units such as velocity (L/T) and force (ML/T²). Table 5-1 show an example of different sets of consistent units. It is good modeling practice to define a specific set of units that will be used in the model by specifying them early in the data deck, as shown in an example MSC/DYTRAN file in Figure 5-2.

Table 5-1 Sets of Consistent Units

Units	SI	English	mm/kg/ms
Length	Meter (m)	Foot (ft)	Millimeter (mm)
Mass	Kilogram (kg)	slug (lbf-s ² /ft)	Kilogram (kg)
Time	Second (s)	Second (s)	Millisecond (ms)
Density	kg/m ³	slug/ft ³	kg/mm ³
Force	kg m/s ² = Newton (N)	slug ft/s ² = lbf	KN
Stress	N/m ² = Pa	(slug ft/s ²)/ft ² =lbf/ft ²	Gpa
Energy	Nm = Joule (J)	(slug ft/s ²)ft =lbf-ft	Joules (J)

This would help the person generating the model, and users downstream that may be involved in editing, debugging or checking the analysis, to quickly recognize and apply the correct input to the model.

Figure 5-2 Example Unit Specification

```

$ SEAT CRASH TEST MODEL
$
$ SI Units: kg - meter - seconds
$ -----
$ conversion factors
$ lbm/in3 to kg/m3: multiply by 2.767990e+4
START
ENDTIME=150.E-3
PARAM, INISTEP, 1.E-6
TLOAD=1
  
```

In general, software such as MSC/DYTRAN, MADYMO or LS-DYNA3D do not require the model to be defined in a particular set of units as long they are consistent. However, careful consideration should be given when the structural finite element model is coupled with an occupant

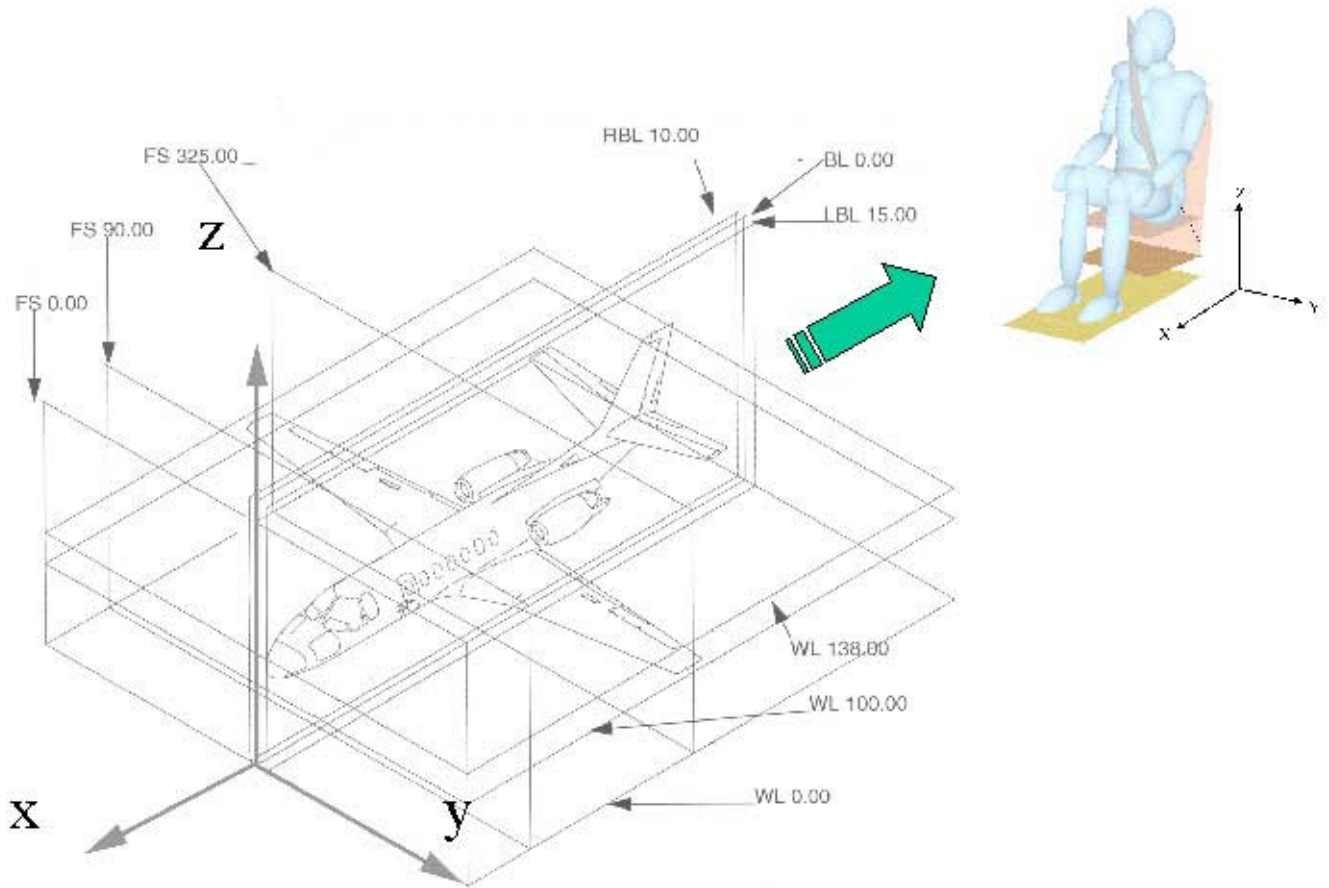
model. For MADYMO the use of SI units with the occupant model is highly recommended due to built-in absolute convergence criteria. Using non-SI units with MADYMO occupants may introduce error in results. Other coupled models - such as MSC/DYTRAN/ATB- will execute well either in English or SI units as long as both the structure and the occupant have consistent set of units.

5.2 COORDINATE SYSTEM

The seat model should be aligned with the aircraft coordinate system. This will facilitate the results of the computer model to be correlated to the test data, where the coordinate and sign convention of the test instrumentation is also oriented in the aircraft coordinate system, as specified in SAE J211. For the seat and sled, the X-axis should be along the fore-aft (fuselage) direction of the aircraft, the Y-axis along the inboard-outboard (buttline) direction, and the Z-axis along the direction of gravity (waterline). Figure 5-3 illustrates a MADYMO model of a forward facing seat aligned in the aircraft coordinate system.

The engineer needs to note the specific orientation of the occupant's axis system, since different occupant models have their own body-attached axis system and may differ from the positive sign convention of the ATD's transducers as specified by the SAE J211 document.

Figure 5-3 Model Coordinate System Orientation



5.3 OCCUPANT MODELS

Most occupant models have been validated for a particular application. For example, the NHTSA Hybrid III occupant model has been extensively validated and used in automotive applications. Cessna has correlated the response of the ATB Hybrid II and MADYMO Hybrid II for aircraft applications with full-scale test data (ref AGATE report C-GEN-3432-1 and C-GEN-3433-1). The ATB Hybrid II and MADYMO Hybrid II occupant models have a response similar to the 14 CFR Part 572 Subpart B Hybrid

II ATD, and therefore are suitable for use in design and certification. Other occupant models may be used for certification if sufficient data is available and the validation task is coordinated with the FAA.

5.3.1 ATB HYBRID II (PART 572 SUBPART B) OCCUPANT MODEL

The ATB Hybrid II (Part 572 Subpart B) occupant model executes within the ATB crash simulation program. Although the ATB program by itself (with multi-body capabilities) can be used to perform crash simulation, the lack of a finite element solver makes it impractical for use in complex analysis and certification where stress results are required. The ATB occupant model is generally coupled with the MSC/DYTRAN finite element codes, although there are current developments to integrate it with LS-DYNA3D within the automotive industry. For practical purposes, this document will provide a brief overview of the ATB HYBRID II model and how ATB is coupled with MSC/DYTRAN. Detailed information of the ATB program, theory or the organization and control of the ATB input deck is available from the ATB Version V Users Manual.

The input for the ATB program is contained in a FORTRAN formatted file with the **.ain* extension (i.e. *seatmodel.ain*). The main output file is identified by the **AOU* extension and contains an annotated listing of the program input and summary of the kinetic energy, accelerations, etc for each requested time step. It is also the primary source for debugging. Tabular time history of specific outputs, such as joint

forces, accelerations and displacements, are generated in the *THS file. Each ATB input file has the following structure as specified in Table 5-2.

Table 5-2 Program ATB Input Card Structure

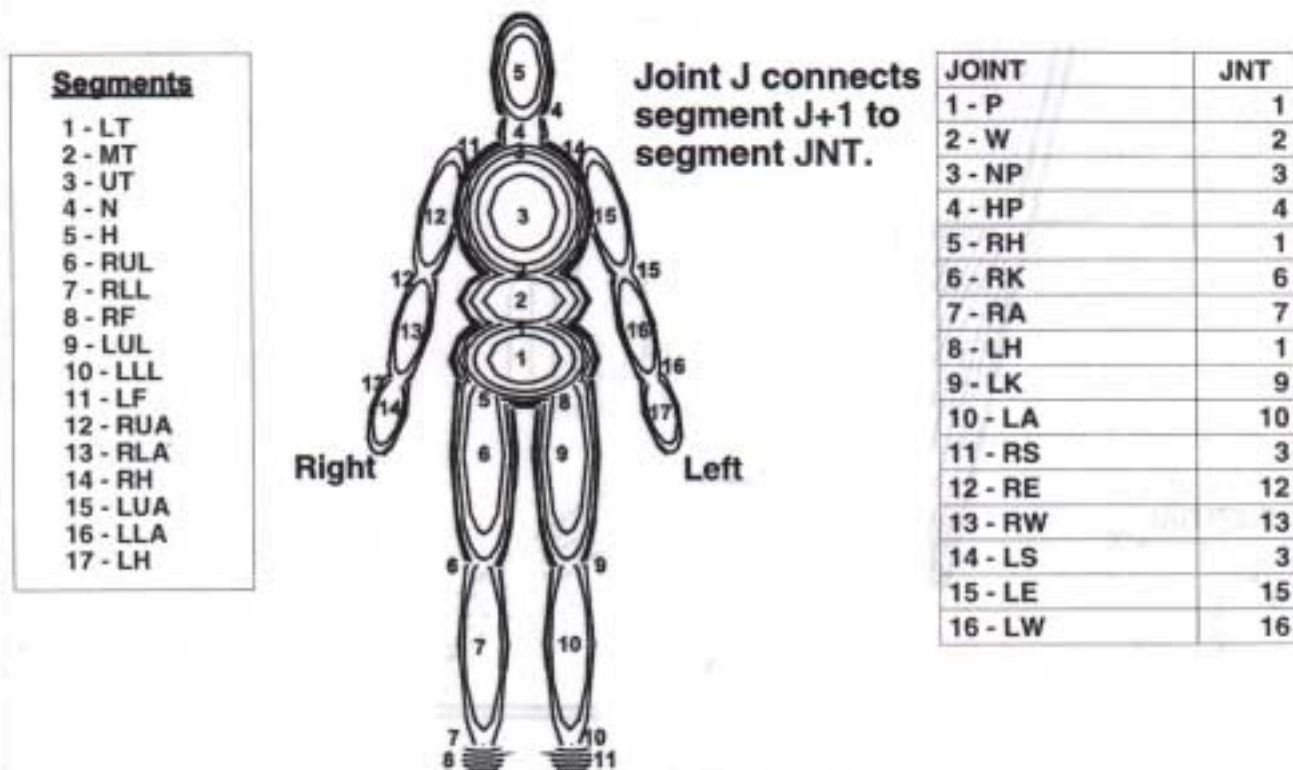
CARD TYPE	DESCRIPTION
Card A.1-A.5	Run control parameters
Card B.1-B.7	Physical characteristics of the body
Card C.1-C.5	Prescribe motion
Card D.1-D.9	Contact surface and other environmental definitions
Card E.1-E.7	Function definitions
Card F.1-F.10	Allowed contacts and associated functions
Card G.1-G.6	Equilibrium constraint assignments
Card H.1-H.12	Tabular time history output control parameters

Definition of each card entry is given in the ATB Model Input Manual. The ATB Hybrid II occupant is comprised of 17 rigid segments connected by 16 pin and spherical joints (Figure 5-4). The geometry, inertial properties and bio-fidelity of the ATB model simulate the NHTSA 49 CFR Part 572 Subpart B ATD. The occupant model is available in English and SI units.

The parent body of the ATB occupant represents the lower torso (Segment 1 - LT). The head acceleration is obtained from Segment 5. Joint number 1 connects the middle torso (MT) to the lower torso (LT), and joint number 2 connects the middle torso (MT) to the upper torso

(UT) of the lumbar column. Therefore, the resultant force in the Z-direction for joint 1 or 2 represents the compressive force of the spinal column.

Figure 5-4 ATB HII Occupant Model



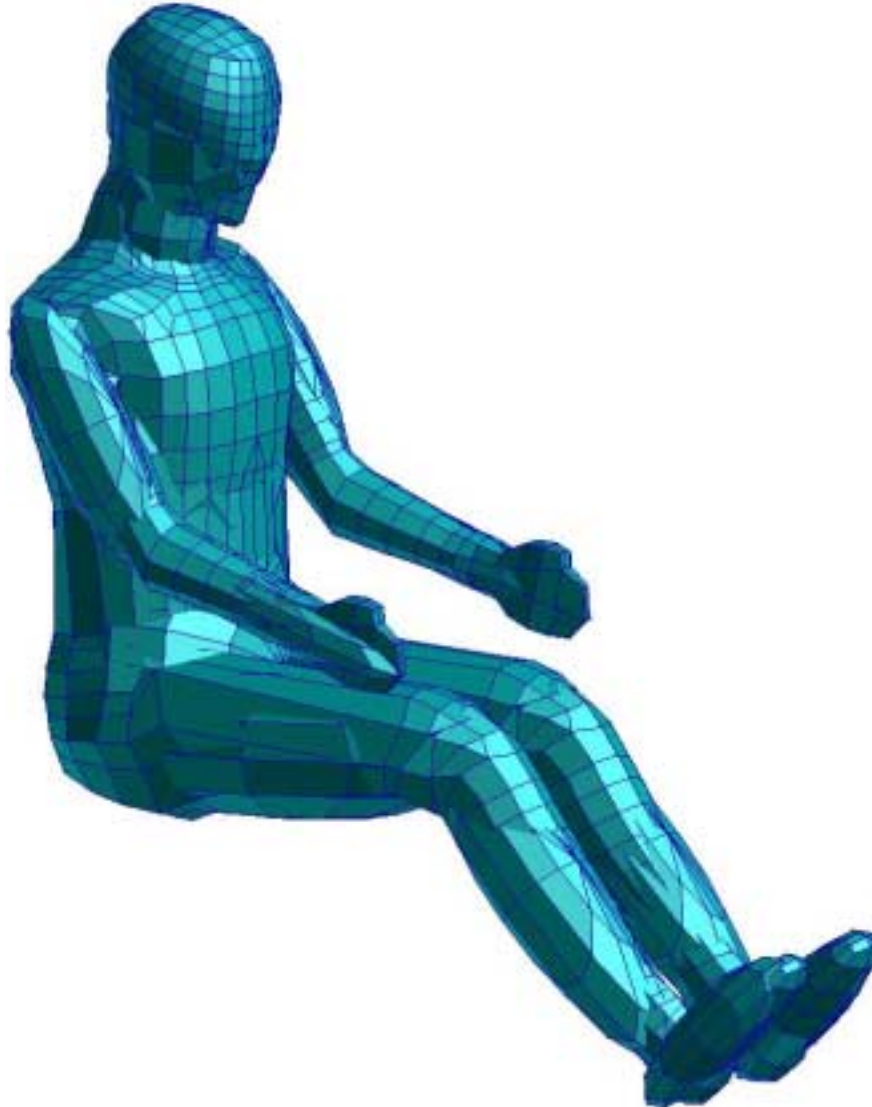
Card G.2 defines the initial position and velocity of the occupant. Orientation of different segments of the body (such as rotating the arms or legs of the occupant) to obtain a desired occupant position is defined by manipulating the coordinate and orientation of each segment in Card G.3.

The ATB model, when coupled with MSC/DYTRAN will appear in the MSC/PATRAN pre/post processor as shown in Figure 5-5. The ATB model was digitized with rigid shell finite elements (with negligible mass)

so that contact with other surrounding finite element structures can be defined. The ATB ellipsoid was coupled to MSC/DYTRAN by means of a RELEX entry. The RELEX entry defines a rigid ellipsoid within the MSC/DYTRAN environment whose properties and motions are governed by ATB. The rigid shell finite elements are then attached to the MSC/DYTRAN ellipsoid through a RCONREL entry, thus completing the finite element definition of the ellipsoid ATB dummy.

Dummy positioning is performed using MSC/PATRAN by running a dummy positioning session file supplied by MSC. The session file enables each individual segment (arms, legs, etc) to be positioned and a new set of nodes will be written out to select the final occupant position. The session file also generates an ATBSEG card, which overwrites the position and orientation of the ATB segments specified in the **ain* file. Since ATB is internally coupled to MSC/DYTRAN, no major change is required to the **ain* input file.

Figure 5-5 Finite Element MSC/DYTRAN ATB Model



5.3.2 MADYMO HYBRID II (PART 572 SUBPART B) DUMMY

The Part 572 Subpart B dummy database available with MADYMO version 5.4 is made of 32 bodies connected with various kinematic joints (reference Figure 5-6). There are seated and standing versions included, but only the seated dummy will be discussed in this

document. See the MADYMO 5.4 Database Manual and User's Manual for detailed information.

Figure 5-6 MADYMO HYBRID II (PART 572 Subpart B) DUMMY

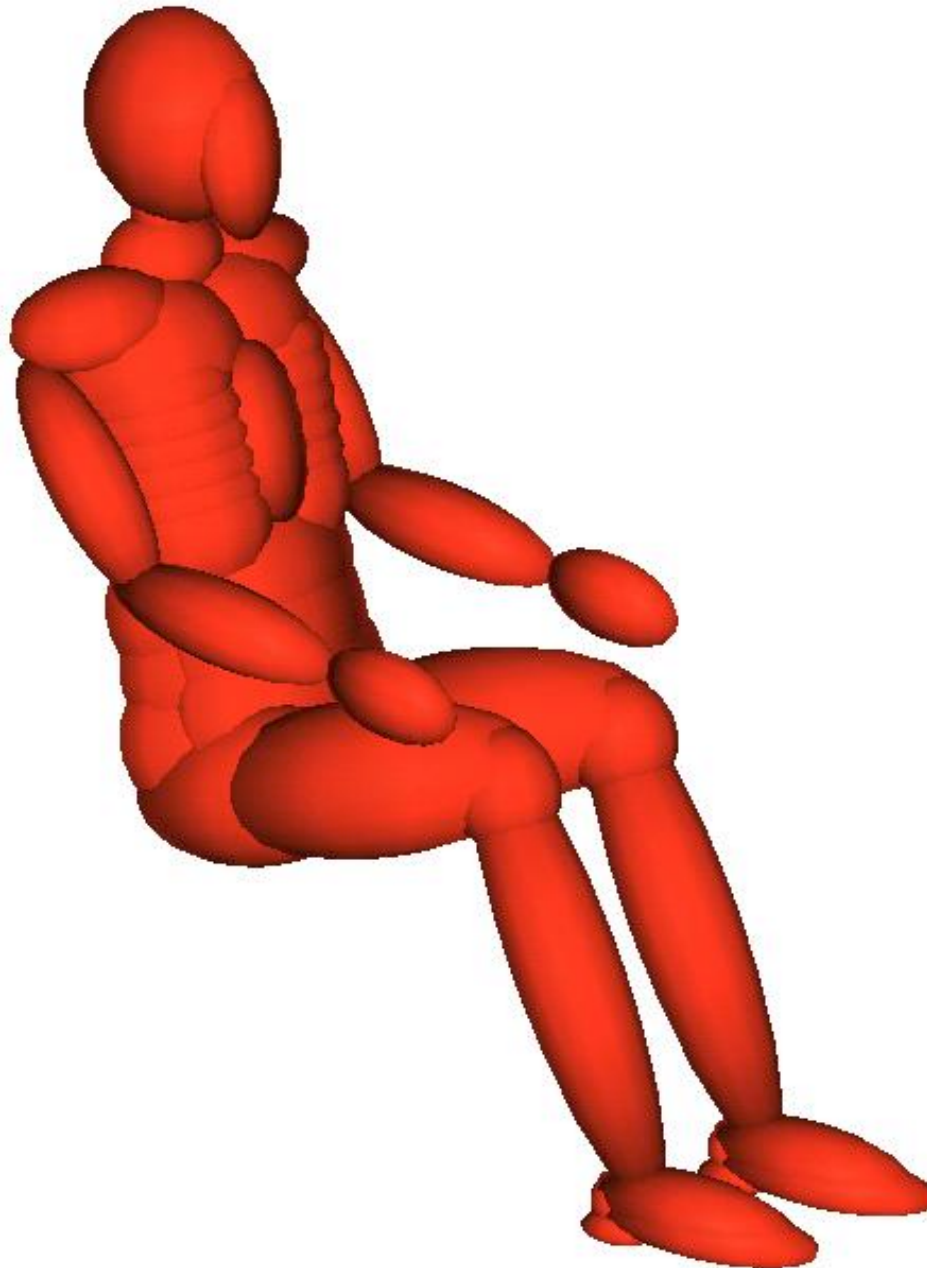


Table 5-3 Standard MADYMO Part 572 Subpart B Dummy Definition

NUMBER	NAME	REMARKS
1	LOWER TORSO	REFERENCE BODY OF DUMMY SYSTEM
2	ABDOMEN	
3	LOWER LUMBAR	
4	UPPER LUMBAR	
5	UPPER TORSO	SPINE BOX AND BACK OF THE RIBS
6	RIBS	FRONTAL AREA OF THE RIB CAGE
7	LOWER NECK BRACKET	FOR NECK ANGLE ADJUSTMENT ONLY
8	LOWER NECK SENSOR	FOR LOAD SENSING ONLY
9	NECK	
10	NODDING PLATE	FOR LOAD SENSING ONLY
11	HEAD	
12	CLAVICLE LEFT	
13	CLAVICLE RIGHT	
14	UPPER ARM LEFT	
15	UPPER ARM RIGHT	
16	LOWER ARM LEFT	
17	HAND LEFT	
18	HAND RIGHT	
19	HAND LEFT	
20	FEMUR LEFT	PROXIMAL OF FEMUR LOAD CELL
21	FEMUR RIGHT	PROXIMAL OF FEMUR LOAD CELL
22	KNEE LEFT	PERIPHERAL OF FEMUR LOAD CELL
23	KNEE RIGHT	PERIPHERAL OF FEMUR LOAD CELL
24	UPPER TIBIA LEFT	ABOVE UPPER LOAD CELL
25	UPPER TIBIA RIGHT	ABOVE UPPER LOAD CELL
26	MIDDLE TIBIA LEFT	IN BETWEEN LOAD CELLS
27	MIDDLE TIBIA RIGHT	IN BETWEEN LOAD CELLS
28	LOWER TIBIA LEFT	BELOW LOWER LOAD CELL
29	LOWER TIBIA RIGHT	BELOW LOWER LOAD CELL
30	FOOT LEFT	
31	FOOT RIGHT	
32	STERNUM	COMPLIANT CENTRAL REGION OF RIB CAGE

The lower torso body is the reference body in the dummy system and connects to inertial space with a free joint (joint number 1), meaning all rotation and translation degrees of freedom are unconstrained. Any other bodies in the dummy system can be traced back to the reference body along a single path (there are no closed loops). Therefore, the overall position and orientation of the dummy is specified by the reference joint degrees of freedom (DOF) entries following the "JOINT DOF" keyword.

The relative orientations of the system child bodies can be adjusted in the input block following the "JOINT DOF" keyword. This allows adjusting the dummy posture from the nominal seated position. Do NOT position the dummy parts by modifying the joint coordinate system orientations in the dummy database following the "JOINTS" keyword, as this will disrupt the joint ranges of motion and stiffness characteristics.

The default dummy database is structured as two trees of keyword/input blocks. The first part of the deck is the system specification enclosed between the keywords "SYSTEM" and "END SYSTEM". The second part of the deck is the output requests enclosed between the keywords "OUTPUT CONTROL PARAMETERS" and "END OUTPUT PARAMETERS". Note that keywords may be abbreviated as specified in the MADYMO Users Manual, for example "SYS" for "SYSTEM" or "END" for "END SYSTEM" etc.

The following entries are in the SYSTEM block:

CONFIGURATION - table defines the body connectivity.

GEOMETRY - defines the coordinates of each joint and joint CG in the parent body coordinate system.

INERTIA - table defines the inertial properties of each body and orientation.

JOINTS - table specifies each joint type, stiffness, and orientation.

INCLUDE - the lumbar spine characteristics are encrypted in the referenced "h350lumb.v03" file.

FLEXION-TORSION RESTRAINTS - defines the force model for the neck and spine.

CARDAN RESTRAINTS - defines the force models for the hips and ribs. Orientations and stiffness functions are specified following this data block.

ELLIPSOIDS - table defines the ellipsoid dimensions, degree, and (optional) contact stiffness characteristics. Orientations and stiffness functions are specified following this data block.

KELVIN - defines a spring-damper element (Kelvin element) for the spine.

CONTACT INTERACTIONS - defines the dummy self-contact evaluations.

POINT-RESTRAINTS - the ribs and abdomen have compressive characteristics defined using point restraints. A point restraint is

equivalent to three mutually orthogonal Kelvin elements. See section 7.3 in *MADYMO Theory Manual Version 5.4*.

JOINT DOF - these values prescribe initial joint position and velocity degrees of freedom.

The following entries are in the OUTPUT CONTROL PARAMETERS section of the dummy model. Additional parameters can be specified as stated in the MADYMO 5.4 User's Manual.

TSKIN - time interval for writing data to kinematic and FE results files.

KIN3 - results format version and options.

TSOUT - time interval for writing data to time history files.

FILTER PARAMETERS - configure signal filters for results data.

LINACC - output requests for linear acceleration vs. time for specified points on bodies, with options to correct for prescribed fictitious acceleration fields.

CONSTRAINT LOADS - output requests for joint constraint loads and filter parameters.

INJURY PARAMETERS - output requests for occupant injury criteria.

Note: The default window size for HIC is set to 36 ms in the MADYMO Part 572 Subpart B dummy model file. The automotive industry uses the 36 ms window. Federal Aviation Regulation's definition of HIC does not

specify a window size other than the full duration of the impact event. 14 CFR Parts 23 and 25 do not explicitly define a time window for HIC calculations, but a maximum window of 50 ms is defined in 14 CFR Parts 27 and 29 (Rotorcraft and Transport Rotorcraft, respectively). In practice, the FAA often imposes the 50 ms maximum window on Part 23 and 25 aircraft certification tests. Automotive regulations (49 CFR 571.208) have recently adopted a 15 ms window with a maximum allowable HIC of 700 for airbag interactions. The modeler should apply the appropriate maximum window based on the impact surface and the negotiated certification requirement.

5.4 MODELING STRUCTURAL ELEMENTS

The modeling of structural elements may consists of the seat structure, cushions, restraint systems, floor structure, instrument panels, glareshields, side panels, crash sled and any other objects that can influence the response of the occupant. There is no ideal method to model structural elements. Generally, each method depends on the capabilities of the software, the information the analyst wants to extract from the model and the desired accuracy of the analysis. There are three basic methods to model structural elements, which are discussed in subsequent sections.

5.4.1 METHOD 1 - MULTI-BODY TECHNIQUES

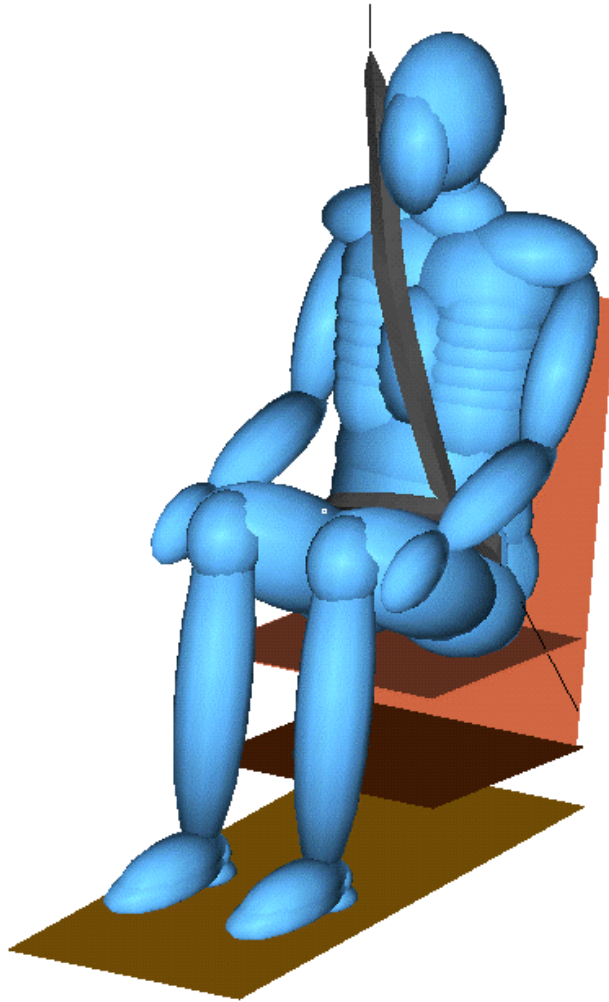
The easiest method to model structural objects is to use multi-body (a Madymo definition) or rigid elements. This includes using combinations of simple planes, cylinders, ellipsoids, and facet

surfaces. Multi-body elements are primarily used to simplify the representation of the structure and are utilized in applications where the kinematic response of the structure is desired but information on stresses and strains are not required. Parts of rigid bodies can be connected together by spring-damper or torsional spring elements to provide resistive force.

Figure 5-7 shows an example of a Madymo seat model generated using multi-body techniques. This model represents an over-spar bench seat, where there is no significant deformation. Because of the rigidity of the seat, structural deformation and stresses were not required. Therefore, a multi-body model is sufficient in determining the response of the occupant.

The seat structure, seat cushion and crash vehicle was represented by multi-body planes. These planes are positioned so that it reflects the correct configuration of the actual seat structure. Each plane has inertial and stiffness properties that are typically obtained from sub-component compression test. For example, the plane representing the seat bottom has a stiffness function that represents the actual seat cushion behavior. The planes are fixed to the vehicle inertial system. Contact is defined between the occupant and planes in terms of load versus deflection.

Figure 5-7 Multi-body model



The model above is particularly useful as a parametric tool because of its simplicity and low computational cost. Changes to the model are easily made, and the next load case is analyzed.

5.4.2 METHOD 2 - FINITE ELEMENT MODELING

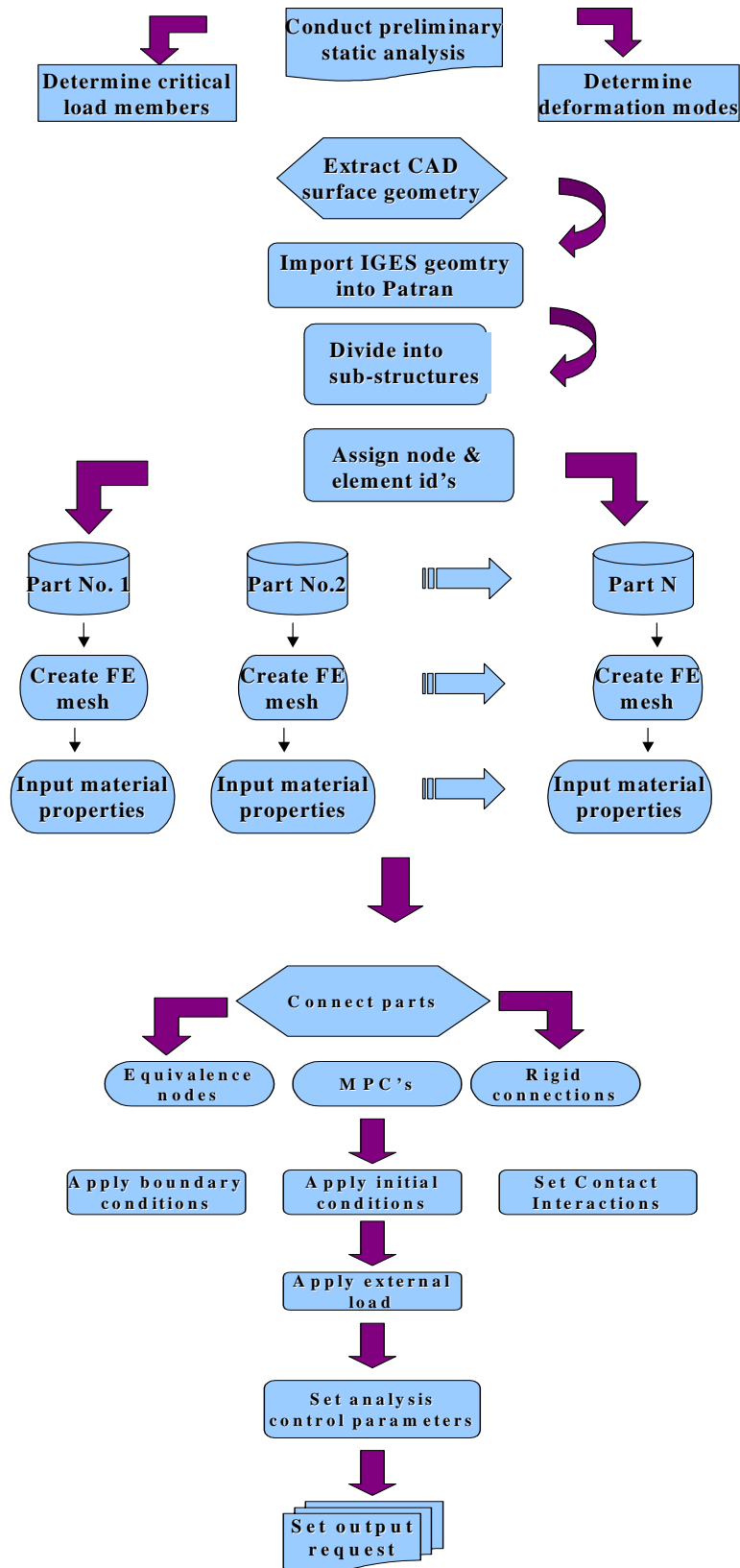
The most representative technique to model structural objects is to use the finite elements (FE) method. FE models are generated to obtain

detailed response of structures and to determine failure modes. Madymo and MSC/DYTRAN have extensive non-linear FE capabilities. Finite element models are more difficult to generate than multi-body models. However, FE models are more practical because they predict realistic structural response and offer the capability to output stresses, strains and internal loads. They can also be utilized to substantiate structural designs. Figure 5-8 shows a process flowchart that is commonly used to generate a FE model.

The precise method on how to generate an efficient FE model of a seat structure will depend on the design of the seat itself and the desired output of the model. Generally, the first step is to determine the load path of the structure for each load case. Then, lists of the critical load carrying members within the load path are noted. Engineering judgment will be used to determine the mode of failure for each critical member. This will help determine the choice of elements to represent the structural.

Geometry data of the seat structure from a CAD package, such as CATIA or Pro-Engineer, is converted to a form such as IGES which can be used by the FE pre-processor as surfaces or solids for generating the FE mesh. Each part of the seat structure is grouped and meshed independently. Care must be taken to ensure that the shape of each element is not distorted in order to avoid computational problems during the analysis. After all components of the structure are meshed, the individual groups are merged by equivalencing coincident nodes.

Figure 5-8 FE Modeling Flowchart



Sometimes, parts can be joined together using spot weld elements or rigid elements. Spotweld elements allow for the joined parts to separate once the loads have met a user defined failure criteria such as tension, shear, torque or moments. Rigid elements are essentially a multi-point constraint (MPC) and are used to define a set of grid points that forms a rigid element.

The next step is to create a database of material properties and assigning them to its respective structure. Pre-processors such as PATRAN have capabilities that link CAD geometry to finite elements, so that the user can select the geometry instead of selecting individual elements (which tends to be more difficult in complex or large size models).

Boundary conditions are applied to constrain the seat model to the vehicle. This is done by selecting the nodes where the seat is attached to the seat rails and applying a constraint to the translational or rotational degrees of freedom. In addition, contact is defined between parts that rest, slide or have the probability of contacting each other.

External loads are applied by prescribing acceleration loads to the seat. The definition of external loads are obtained from the actual crash sensor or the analyst can apply a fictitious triangular pulse prescribed in the FAR's.

The final step in the process includes setting analysis control parameters such as analysis termination time, integration method,

hourglass energy control, mass scaling and selecting the set of output request. At this point , the model is ready to be executed. However, rarely does a FE model execute flawlessly during the first attempt. The process typically goes through a cycle of error debugging and correction of the input deck.

5.4.2.1 ELEMENT TYPES

There are many types of finite elements, and the choice of element selection will depend on the load and deformation characteristics of the actual structure that it will represent. Finite elements used for structural analysis are also known as Lagrange elements in terms of the formulations of these elements. Finite elements are typically grouped in the following categories:

1. Scalar Elements

Typically consist of spring, mass and damper elements. The stiffness properties are usually user defined. Scalar masses are commonly used to model a concentrated mass at one location, such as an engine block, fuel contents or ballast weights. There is no stiffness definition associated with a scalar mass.

Figure 5-9 Spring Element



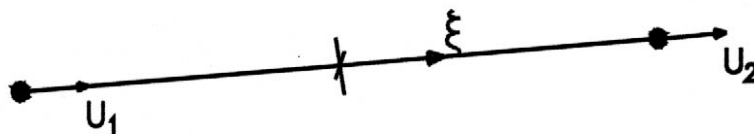
Spring elements connect two grid points and the force acts in the direction of the connecting grid points. Spring elements connect translational and rotational degrees of freedom and may have linear or non-linear stiffness property. For translational springs, the stiffness is defined in terms of force versus deflection. For rotational springs, the stiffness is defined in terms of moment versus angle of rotation.

2. One-dimensional elements

One-dimensional elements are used to represent structural members that have stiffness along a line or a curve. Examples of one-dimensional elements are rod and beam elements.

Rod elements carry tension and compressive loads only. The mass of the elements are lumped and distributed equally at the nodes. The only geometry property required is the cross-sectional area of the rod.

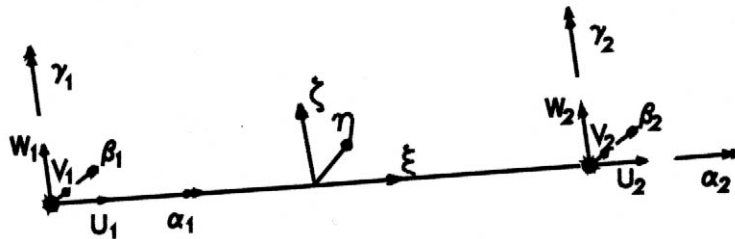
Figure 5-10 Rod Element



Beam elements carry axial, torsion and bending loads. The mass of the beam is lumped and equally distributed over the two nodes. Care has to be taken regarding the center of mass, shear center and centroid definition of the beam definition for each code. Unless otherwise stated, the mass, shear center and centroid of the cross-section all

coincide. The orientation of the beam should be defined in its element coordinate system. The geometry property required are the area and moments of inertia of the beam.

Figure 5-11 Beam Element



3. Two-dimensional elements

Consist of membrane, quadrilateral and triangular elements. These elements are most widely used because of the versatility and robust formulations. The mass of two-dimensional elements are lumped and equally distributed over all the nodes.

Membrane elements carry in-plane loads and do not have bending stiffness. Membrane elements can be three or four-node elements with three translation degrees of freedom on each node. The deformation is determined by the translation degrees of freedom on these nodes. Depending on the code, membrane elements can have linear or non-linear properties. Seat belt webbing and seat pans are modeled using membrane elements. The geometry property required is the thickness of the membrane.

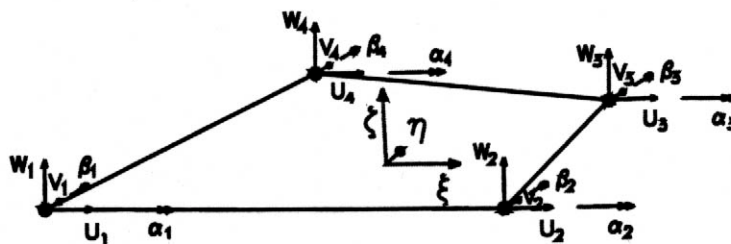
Quadrilateral shell elements are the most widely used. They carry in-plane as well as bending loads. Shell elements have six degrees of freedom at each node; three translations and three rotations.

Transverse shear stiffness is accounted for by a shear correction factor. The geometry properties required are the shell thickness and the number of integration points through the thickness.

Triangular elements typically exhibit a stiffer response and are used only as transitional elements and in areas of low stress concentrations.

Most codes use a default one-point integration at the center of the element, although there are options to increase the number of integration points at the expense of computational efficiency. Note that when one-point integration is used, the hourglass or zero energy modes are generated and which will have to be suppressed.

Figure 5-12 Shell Element



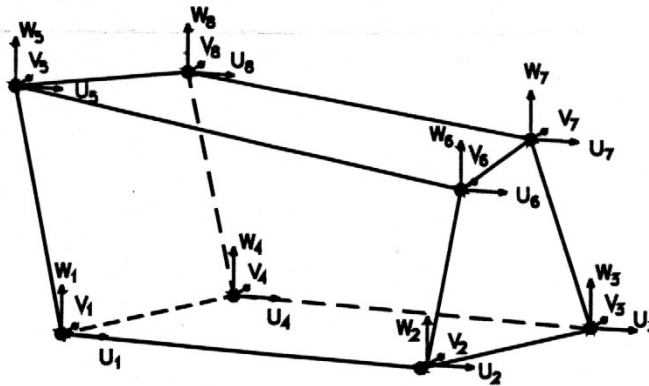
Over the years, advanced formulations have allowed for more robust and computationally efficient elements, such as the Belytschko-Tsai, Key-Hoff and Hughes-Liu shells. The choice of element formulation usage will depend on the need to compromise accuracy with computational

speed. In most cases, the Belytschko-Tsai shell formulation would suffice.

4. Three-dimensional elements

Three-dimensional elements are also known as solid elements and consist of tetra, penta and hexa elements. The element is capable of carrying tensile, compression and shear loads. The mass of the solid elements are lumped and equally distributed over all nodes.

Figure 5-13 Solid Element

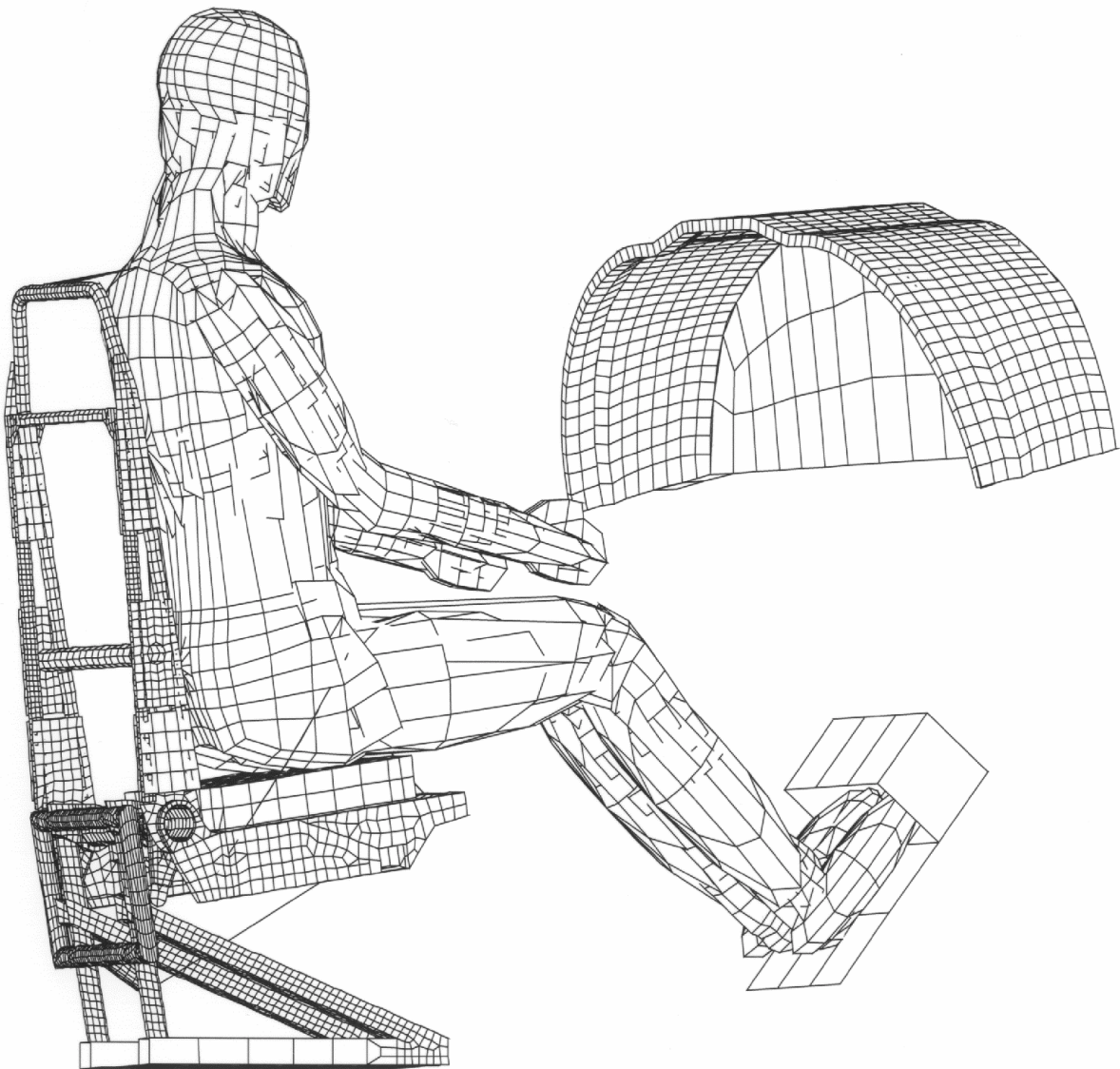


The hexahedral element is commonly used because of its efficiency, and it is easier to mesh and interface with other elements. The tetra and penta elements are degenerated forms of the hexa elements where the grid points coincide resulting in significant reduction in performance. The solid elements use one-point (or reduced) integration for computational efficiency. However, this also results in twelve zero energy or hourglass modes. These modes will have to be suppressed using the hourglass energy control parameter that is available in all codes.

5.4.2.2 EXAMPLE FE MODEL

Figure 5-14 shows an example of an MSC/DYTRAN FE seat model. The purpose of the model was to obtain an accurate prediction of the structural response, locate areas of high stress concentrations and determine how the seat affects the occupant's trajectory.

Figure 5-14 MSC/DYTRAN FE Model

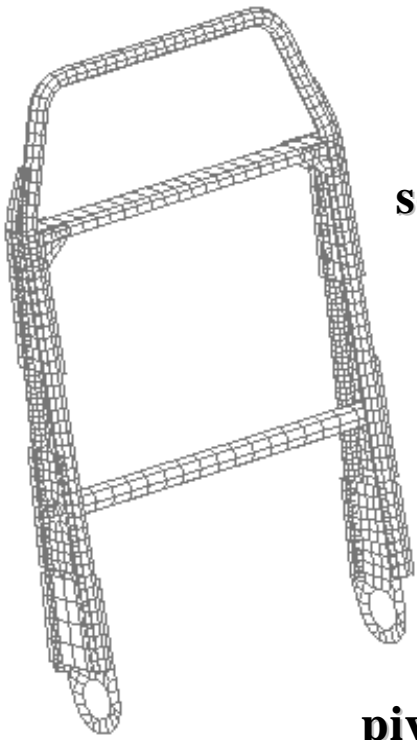


Shell elements were the most widely used of all Lagrange elements because of its robust formulation and versatility. The seat assembly consists of five (5) primary structures; seat back, seat bucket, seat pan, seat base and seat pivot assembly.

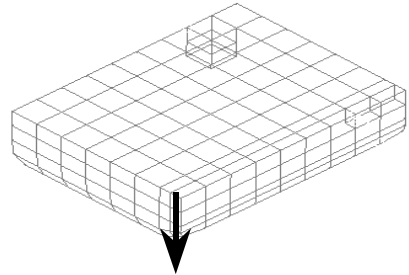
The seat was modeled in the aircraft coordinate system consistent with the definitions presented in Section 4.2. The crash load from the occupant is transferred to the seat from the anchor points on the shoulder harness and the lap belt. In the forward impact case, the load is transferred from the seat back down to the pivot mechanism, and finally to the diagonal cross members on the seat base in the form of compression load.

The majority of the seat structure was modeled using 4-noded quadrilateral CQUAD4 (KEYHOFF formulation) shell elements. Triangular CTRIA3 (CO-TRIA formulation) elements were used as transition elements in non-critical stress areas. Seat adjustment mechanisms of structural significance - such as Hydroloks and recline arms - were modeled using non-linear spring and simple beams elements. The seat cushion was modeled using CHEXA solid elements with equivalent cushion thickness. The footrest and sled is modeled using CQUAD4 elements using rigid (MATRIG) material properties. Figure 5-15 shows an exploded view of the finite element structure.

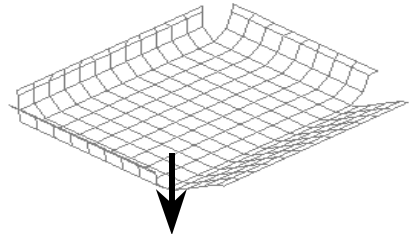
Figure 5-15 Exploded View of FE Seat



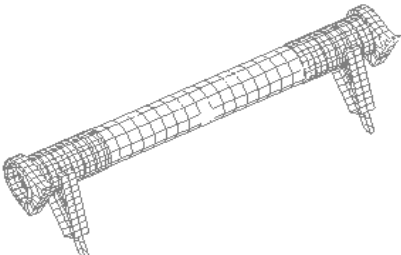
seat back



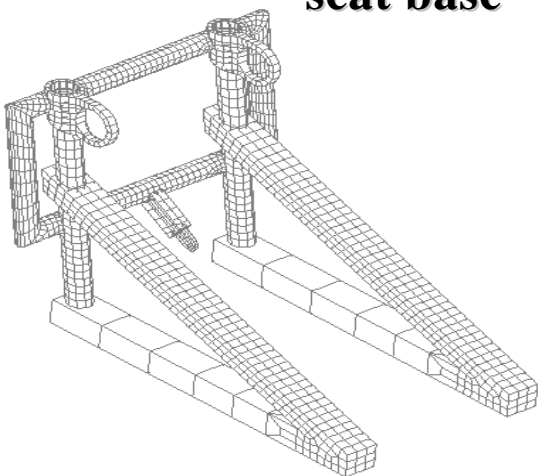
seat cushion



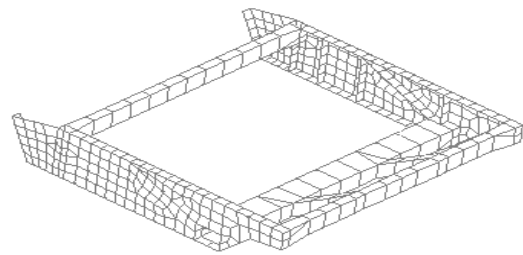
seat pan



pivot assy



seat base



seat bucket

5.4.3 METHOD 3 - HYBRID MODELING METHOD

Multi-body and finite element techniques can be combined to model structures. This is a common method used in Madymo (although the same method can be applied in MSC/DYTRAN using the rigid ellipsoid capabilities). The hybrid method is used to simplify the FE modeling process, replacing non-critical FE elements with multi-body ellipsoids or planes.

Figure 5-16 MADYMO Hybrid Modeling Model

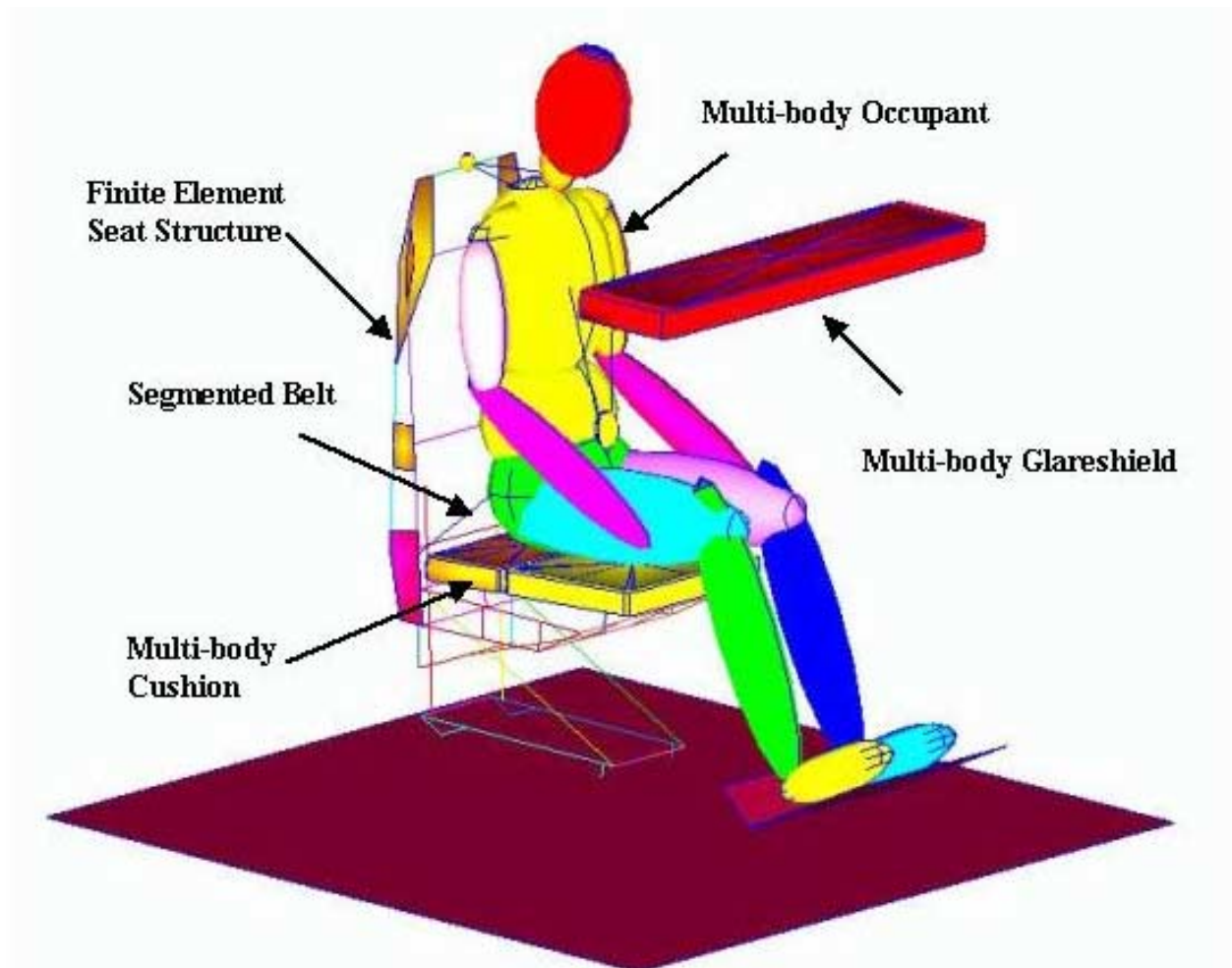


Figure 5-16 shows an example of the hybrid modeling techniques used to model the same seat in Section 5.4.2. The seat frame was modeled using a combination of beam and shell elements. However, the seat cushion and glareshield was modeled using two ellipsoidal multi-body elements instead of finite elements (as oppose to the finite element cushion in Figure 5-15). In this case, sub-component test must be conducted to obtain the load-deflection characteristics of the seat cushion and glareshield to charactrize its response during impact.

The seat cushion ellipsoids are rigidly connected to the seat bucket at its corner locations using FE-to-multibody constraints. The glareshield is fixed in inertial space. Loads, constaints and boundary conditions are applied in the same manner as the FE model.

In general, hybrid models are less accurate than FE models. The hybrid model uses less CPU resource than a full FE model and is sufficient to predict with reasonable accuracy the deformation of the seat structure and the response of the occupant.

5.4.4 MODELING FAILURE OF JOINTS OR FASTENERS

There are numerous methods of simulating structural failures in a nonlinear finite element model. A typical failure mode modeled in seat analysis is failure of rivet and threaded fastener joints. MADYMO, LS-DYNA, and MSC/DYTRAN have capabilities to model simple shear and tensile failure of fasteners. More complex continuum damage mechanics (CDM) material failure models are available for structures

modeled with these codes. As an example, rivet failure can be modeled using MADYMO node-to-node spotweld constraints as shown

```

SPOTWELDS
  NODE-NODE
    MAXFN          300.0
    AN              2.0
    MAXFS          350.0
    AS              2.0
    WINDOW         0.001
    1 743 2 21      FEMHIS
    1 632 2 110     FEMHIS
    1 1219 2 35     FEMHIS
  END NODE-NODE
END SPOTWELDS
  
```

This example defines three node-node spotwelds in MADYMO 5.4 format. The spotwelds are defined as having a maximum allowable normal force of 300.0, and a maximum allowable shear force of 350.0. The failure criteria is defined as follows:

$$\left| \frac{F_{NORM}}{MAXFN} \right|^{AN} + \left| \frac{F_{SHEAR}}{MAXFS} \right|^{AS} < 1$$

The shear and normal failure criteria exponents are set to 2. These exponents determine the rupture criterion shape. The time window (0.001) specifies the time duration that the failure criteria must be violated before the failure initiates. The spotwelds are defined between nodes in FE models 1 and 2, node pairs: 743 and 21, 621 and 110, and 1219 and 35. A vector between the nodes of a spotweld must have a magnitude greater than zero. The optional FEMHIS keyword in the example requests output of the shear and normal forces.

5.5 RESTRAINT MODELING

Restraint modeling techniques are presented for the most common restraint configurations used in FAR Part 23 type aircraft: forward, side and aft facing, 2-5 point restraints. These restraints are almost always composed of two inch nylon or polyester webbing. The belt ends attach to the seat or airframe with a pin joint or an inertia reel/webbing retractor, and are joined together with a metallic buckle on the lap belt.

5.5.1 METHODS

There are three possible methods of modeling belt systems:

- segmented belt model (spring-damper segments).
- finite-element model (membrane or truss elements).
- hybrid model combining segmented belts and finite-elements.

ATB and MADYMO offer segmented belt models. LS-DYNA3D, MADYMO, and MSC/DYTRAN have finite elements suitable for restraint modeling. MADYMO has hybrid restraint modeling capability.

5.5.1.1 SEGMENTED BELT MODEL

The segmented belt (available in ATB and MADYMO) is a simple restraint model represented by linear segments with user defined nonlinear spring-damper characteristics including hysteresis for unloading and reloading. Initial slack or tension can be assigned to belt segments. The belt ends can be optionally defined as retractors /pretensioners. The belt segments are attached to the occupant at various points. Belt

segments allow slip along the length of the belt, but not transversely. The lack of lateral slippage may reduce the accuracy of the simulation and belt loads in some cases. The segmented belt is suitable to simulate occupant restraint and predict tensile loads where there is minimal expected transverse slippage.

The webbing retractor option of the segmented belt model can simulate pay-out, locking, and pre-tension of a production inertia reel or retractor. The MADYMO segmented belt model can be locked based on user specified sensor signals including vehicle acceleration and belt feed rate. Specify the appropriate locking criteria and the force-deflection characteristics of the device being modeled.

5.5.1.2 FINITE ELEMENT RESTRAINTS

A finite element belt offers the best contact model, including transverse sliding of the webbing on the occupant and seat model. The model requires the following inputs:

- A discrete mesh of the restraint geometry in the pre-test position.
- Material properties appropriate for the magnitude of loads to be applied.
- Element properties (cross sectional area or thickness, and formulation)
- Boundary conditions (contact, belt connectivity, supports).
- Friction characteristics (static and dynamic coefficients or friction function)

The mesh should be generated to represent the correct belt geometry as applied to the seated occupant. Contact evaluation must be defined between the nodes or elements and the occupant. Webbing material properties can be obtained from tensile tests.

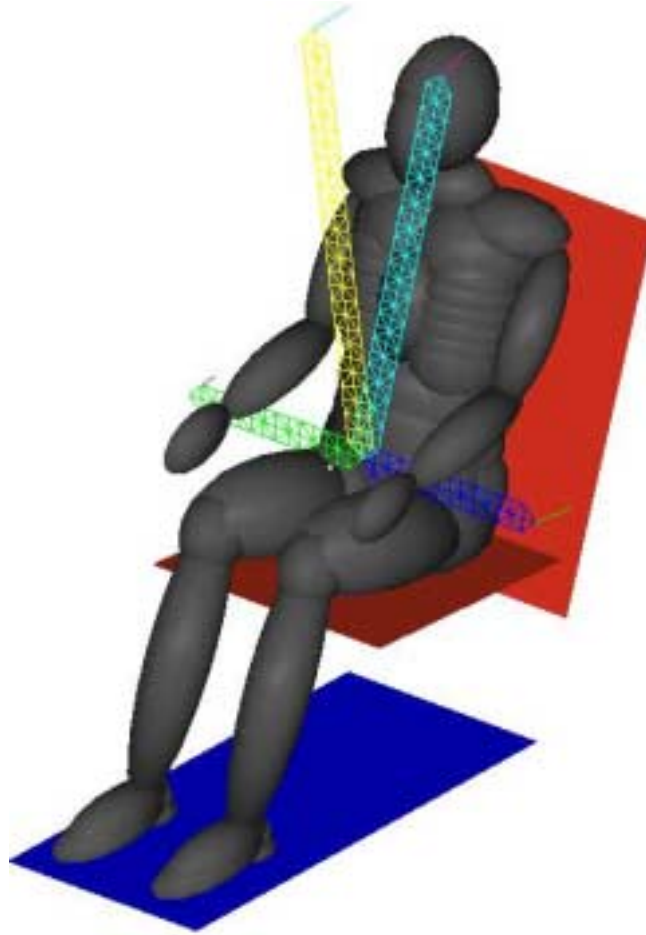
In MADYMO, the recommended 2-d element for belt webbing is the MEM3NL (plane, constant stress triangular elements with in-plane and no bending stiffness) membrane element with HYSIS0 material (elastic isotropic material with hysteresis). The recommended 1-d element in MADYMO is the TRUSS2 element (uniaxial with tension and compression stiffness) with HYSIS0 material definition. For LS-DYNA3D, use the 1-d *ELEMENT_SEATBELT with *MAT_SEATBELT (belt webbing material) and *SECTION_SEATBELT (defines a seatbelt part). LS-DYNA3D also has webbing retractor, pretensioner, accelerometer, sensor, and slipping belt options. In MSC/DYTRAN, use the CROD element with PBELT properties.

5.5.1.2.1 PRESIMULATION

It is difficult to manually generate an FE mesh of an applied belt restraint. Finite element belt models typically require a pre-simulation analysis to obtain the initial nodal coordinates of the belt applied to the occupant. For a membrane belt model, create a flat mesh of the belt webbing and position the segments near the target location on the dummy (Figure 5-17). Use a linear-elastic material with stiffness considerably higher than the actual belt stiffness. This increased stiffness reduces element distortion during the presimulation. Define contact between the belt nodes and the dummy.

Lock the joints to prevent the dummy from moving out of the test position.

Figure 5-17 MADYMO 4-Point Restraint Before Pre-simulation



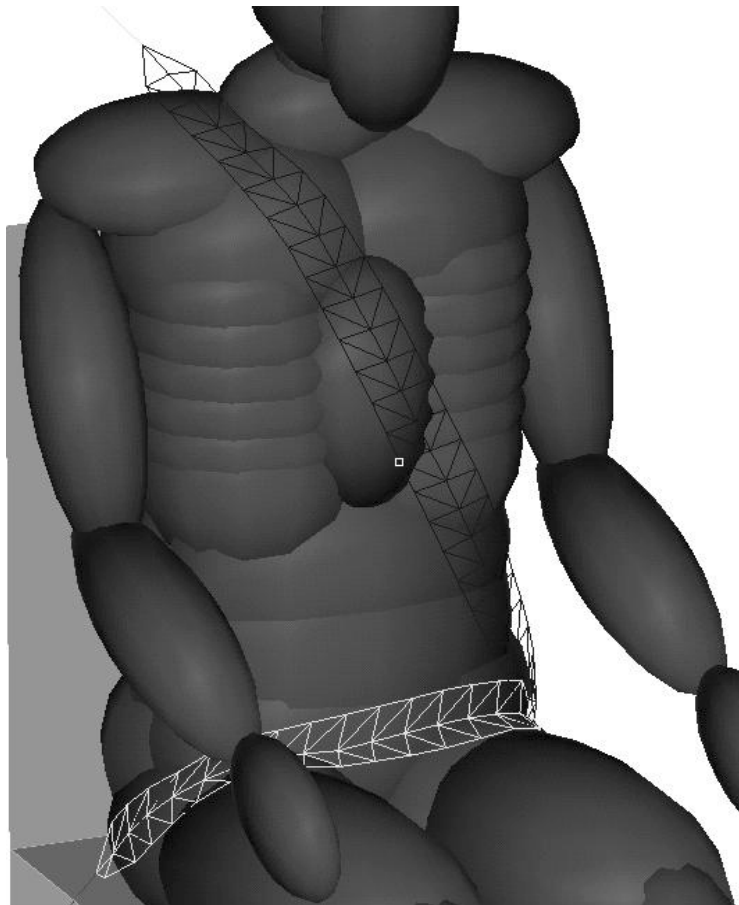
There are various methods of applying the belt to the occupant in each code. In MADYMO, attach belt segments to the end nodes of the belt mesh, and apply pretensioners to pull the belts in to place. In MADYMO and other finite element codes, another method to move the belts into position is to specify a nodal displacement versus time for the belt end nodes. Request nodal coordinate output from the presimulation analysis and use the coordinate output to initialize the positions for the final simulation. Before running the impact analysis, replace the

presimulation material properties with appropriate values, and unlock the dummy joints.

5.5.1.3 HYBRID BELT MODEL

The MADYMO hybrid belt model is simply a finite element belt model combined with segmented belts. The finite element part is usually modeled as the portion of the webbing that contacts the occupant. The segmented belts usually connect the end nodes of the FE belt to the airframe or seat (see Figure 5-18). Use the segmented part of the hybrid belt to model retractors, pretensioners, and sliprings.

Figure 5-18 MADYMO Hybrid Belt After Pre-simulation



5.6 MATERIAL MODELS

The selection and use of appropriate material models is critical in determining the accuracy of the analysis. The formulation of advanced material models have increased significantly, and most explicit codes have numerous material models even for a simple material such as aluminum. This makes selecting the correct material models confusing. For example, LS-DYNA has over 100 material models, with eleven different material models to treat the behavior of foam materials. Thus, a comprehensive discussion on all the available material models is not practical. However, this section will attempt to provide some guidance on the selection of material models that have been shown to be effective for the analysis of components that are commonly used in seat structures.

The first thing to remember in selecting material models is to begin the analysis with simple material models such as an elastic material model. The reason is that simple material models make debugging easier during the initial stages of the analysis, and allow the program to execute without introducing additional errors. Also, the response of elastic material models is easier to comprehend. The analyst can view the results, and then make a determination if a more complicated elastic-plastic model is required to further enhance the accuracy of the analysis.

Complicated material models require specific inputs and coefficients that are obtained by conducting special tests. For analysis performed

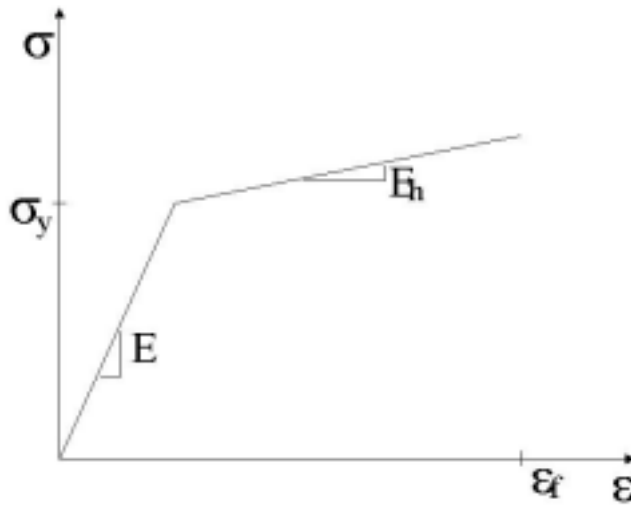
with MADYMO multi-body techniques, component test are required to obtain the load versus deflection characteristics of each structure. User manuals provided with the software provide the descriptions of each material model and suggest applications associated with it.

5.6.1 METALLIC MATERIAL MODELS

Material models associated with aircraft seat metallic structures such as aluminum and steel come with a variety of formulations. These materials are considered isotropic (the an-isotropic behavior of thin sheet metals can be neglected as they exhibit such behavior primarily in high-velocity impact sheet metal-forming applications and do not represent the behavior of the same material in seat analysis).

The simplest is the elastic material model, which describes a linear relationship between the six stress and strain components. Elastic material model input requires only two material constants: Young's modulus E and Poisson's ratio ν . If the material is expected to yield under crash loads, an elasto-plastic model can be used (Figure 5-19). In this case, the material will undergo linear elastic and linear plastic strain (bilinear or piecewise linear). In addition, most codes will also allow for a failure strain value to be defined.

Figure 5-19 Elasto-Plastic Material Model



The elastic constants, along with the yield stress, material density and hardening modulus can be obtained from MIL-Handbook-5. Linear properties for Aluminum 2024-T3 and 4130 Steel are listed in Table 5-4.

Table 5-4 Input Data For Aluminum 2024-T3 and 4130 Steel

Material	Young's Modulus (psi)	Poisson's Ratio	Yield Stress (psi)	Density (lbf-s ² /in ⁴)
Aluminum 2024-T3	1.05E+7	0.3	48000	0.0002621
Steel 4130	2.90E+7	0.32	75000	0.000741

Table 5-5 shows a matrix of cross-reference for the different elastic and elasto-plastic models for the different codes. These input cards

are valid for shell elements, and the user needs to reference the user's manual for the appropriate material card for other elements.

Table 5-5 Matrix of Material Models for Metallic Structures

MATERIAL MODEL	Analysis Code		
	MSC/DYTRAN	LS-DYNA3D	MADYMO
Elastic	DMATEL	*MAT_ELASTIC	ISOLIN
Elastic-Plastic	DMATEP	*MAT_PLASTIC_KINEMATIC	ISOPLA

For some metals, such as mild steel, the material yields at a higher effective stress state at increased strain rates. The strain rate sensitive behavior of steel has significant benefits for crashworthiness applications as it increases the stiffness of the structure under crash loads. The strain rate hardening law is formulated as

$$\sigma_d = \sigma_{y_0} g(\dot{\epsilon}) + \sigma_{y_1} (\epsilon_p)$$

where σ_{y_0} is the initial yield stress, g the strain rate dependency function and ϵ_p is the effective plastic strain. The strain rate dependency function is treated using the Cowper-Symonds strain rate empirical function

$$g(\dot{\epsilon}) = 1 + \left(\frac{\dot{\epsilon}}{c_1} \right)^{c_2}$$

where C_1 and C_2 are strain rate enhancement coefficients. In MSC/DYTRAN, the rate effects are modeled using the DYMAT24 material card. An example input card for 4130 steel (in English units) is

shown in Figure 5-20 where the C_1 and C_2 are strain rate enhancement coefficients are 40.4 and 5.0 respectively.

Figure 5-20 Example MSC/DYTRAN Input for Strain Rate Material

```

$ ----- Material 4130_steel_solid id =7
DYMAT24      7 .000741 2.9e+07   .32
+A000509
+A000510    75000                0.37   40.4   5.0           +
+
+                               DYNA
```

The rate effects can also be modeled using the MAT_PLASCTIC_KINEMATIC material input card in LS-DYNA3D where the SRC and SRP input card represents the C_1 and C_2 strain rate enhancement coefficients. An example input deck is provided in Figure 5-21.

Figure 5-21 Example LS-DYNA3D Input for Strain Rate Material

```

*MAT_PLASTIC_KINEMATIC
$      MID      RO      E      PR      SIGY      ETAN      BETA
      2  0.000741  2.9+7    0.32    75000    176000     0
$      SRC      SRP      FS
      40.4      5.0      .37
```

In MADYMO, the strain rate effects for steel can be modeled using the ISOPLA material card. An example input format (in SI units) is shown in Figure 5-22

Figure 5-22 Example MADYMO Input for Strain Rate Material

```

MATERIALS
* CollectorName>> 4130N steel
TYPE ISOPLA
E 2.000E+11
DENSITY 7915
YIELD STR 5.172E+08
RATE DEP COWPER
DRATE 40.4
PRATE 5.0
```

5.6.2 COMPOSITE MODELS

Laminated fiber-matrix materials such as fiberglass-epoxy are frequently used in aircraft seats, glareshields, side ledges, cabinets, and tables. Finite element codes like LS-DYNA3D and MSC/DYTRAN typically include material models for composites. Laminated structures with or without a core can be modeled using a shell element mesh. Techniques for modeling composite structures are described in the following sections.

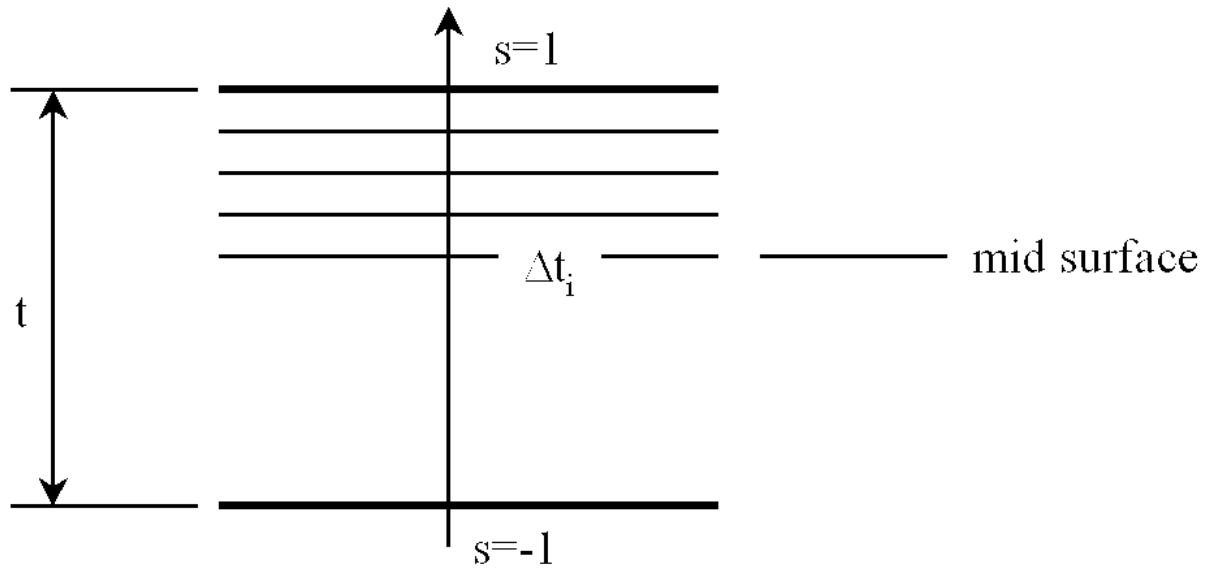
5.6.2.1 MODELING COMPOSITES WITH LS-DYNA3D.

There are several options for modeling layered composites with shell elements. The simplest and least general is to use the BETA option of *Section_Shell to define the material direction for each integration point through the element thickness. A user-defined integration rule should also be used to control the layer thickness. (see IRID of *section_shell, and integration_shell). If a composite is made up layers of different materials, a more general composite can be modeled by specifying a different part ID for each integration point (see *integration_shell). Each part can refer to a different material model with the restriction that all materials must be of the same type.

For example, you could specify an element with one layer of material type 2 using $E_a=10$, $E_b=1$, and another layer of material type 2 with $E_a=3$, $E_b=3$, where E_a and E_b are the Young's modulus in the 'a' and 'b'

directions. This method allows different material constants to be used in the different layers, but still does not allow complete general mixing of material types in a single shell element.

Figure 5-23 User Defined Shell Integration Points



For complete freedom of mixing materials in a composite, it is necessary to model the section with multiple elements, one element for each material type in the composite. The elements should all be given a thickness equal to the total composite thickness and should all share the same nodes, so they would appear to all lie in the same space when viewed in a preprocessor or postprocessor. However, in order to obtain the correct membrane and bending stiffness for the whole composite element, define a separate integration rule for each element in the composite with appropriate weights and through thickness locations.

A simple example of a sandwich type composite with one material in the middle and another on the top and bottom surface might have integration rules like this. The middle material could have 2 integration points with weights and thickness coordinates of

$$Wf1 = 0.25, S1 = -0.25$$

$$Wf2 = 0.25, S2 = +0.25$$

The surface material could have 2 integration points with weights and thickness coordinates of

$$Wf1 = 0.25, S1 = -0.75$$

$$Wf2 = 0.25, S2 = +0.75$$

In this example, Wf1 is the weight factor for integration point 1, and S1 is the thickness direction coordinate of integration point 1 etc. The correct stiffness is achieved so long as the total weight of all elements is equal to 1, and the thickness coordinates are defined such that the integration points are at the middle center of each layer. This method has complete freedom of material type for each layer. It seems like a great idea, but LS-DYNA has built in protection to prevent the input of weights that don't add up to 1. If you try the example, it converts the weights at all integration points to 0.5 so that they add up to one for each element. Fortunately, there is a trick to get around this protection.

The idea is to reduce the thickness of elements accordingly so that the correct membrane stiffness is achieved for each material. In other words, each element should have a thickness equal to the actual

summed thickness of layers of that material. In the example, the element thickness of both elements should be reduced to one half the total composite thickness. To achieve the correct bending stiffness for the composite, the thickness coordinates for each integration point should be increased accordingly. In the example, since each element has been reduced to half the composite thickness, the thickness coordinates should be doubled, so $S1=-0.5$, $S2=+0.5$ for the middle material, and $S1=-1.5$, $S2=+1.5$ for the surface material. Notice that this violates the usual restriction that the thickness coordinates should be in the range of -1 to +1. However, it works because LS-DYNA does not enforce this rule.

In this simple example, each of the 4 material layers has a thickness of 1/4 of the total element thickness. However, there is absolutely no restriction on the number of layers, thickness of layers, or material of the layers using this method. The only rules that should be followed to achieve correct stiffness of the overall composite element are:

1. The sum of individual element thickness should equal the total composite thickness.
2. The thickness coordinates for each integration point should be multiplied by the total composite thickness and divided by the corresponding element thickness.

If multi-element method is used, care should be taken if the composite is to be checked for contact. Only one of the elements making up the composite should be checked for contact since all elements share the

same nodes. However, the element thickness will be less than the composite thickness, so it may be desirable to directly prescribe the thickness for contact (see SST,MST on *contact) if using a contact type where element thickness is taken into account.

5.6.3 SEAT CUSHION FOAM MODELS

The selection of the appropriate foam model for modeling seat cushion is critical in obtaining accurate spine load prediction. Foam models are typically formulated for a particular type of foam behavior. For instance, LS-DYNA3D has different material models for commonly used aircraft seat cushion foam such as DAX (polyurethane) versus slow recovery foams such as Confor. Other codes such as MADYMO rely on the user to obtain the specific load versus deflection response of the foam from a component level test as input data for the foam material model. The following is an example of seat cushion foam modeling using MSC/DYTRAN.

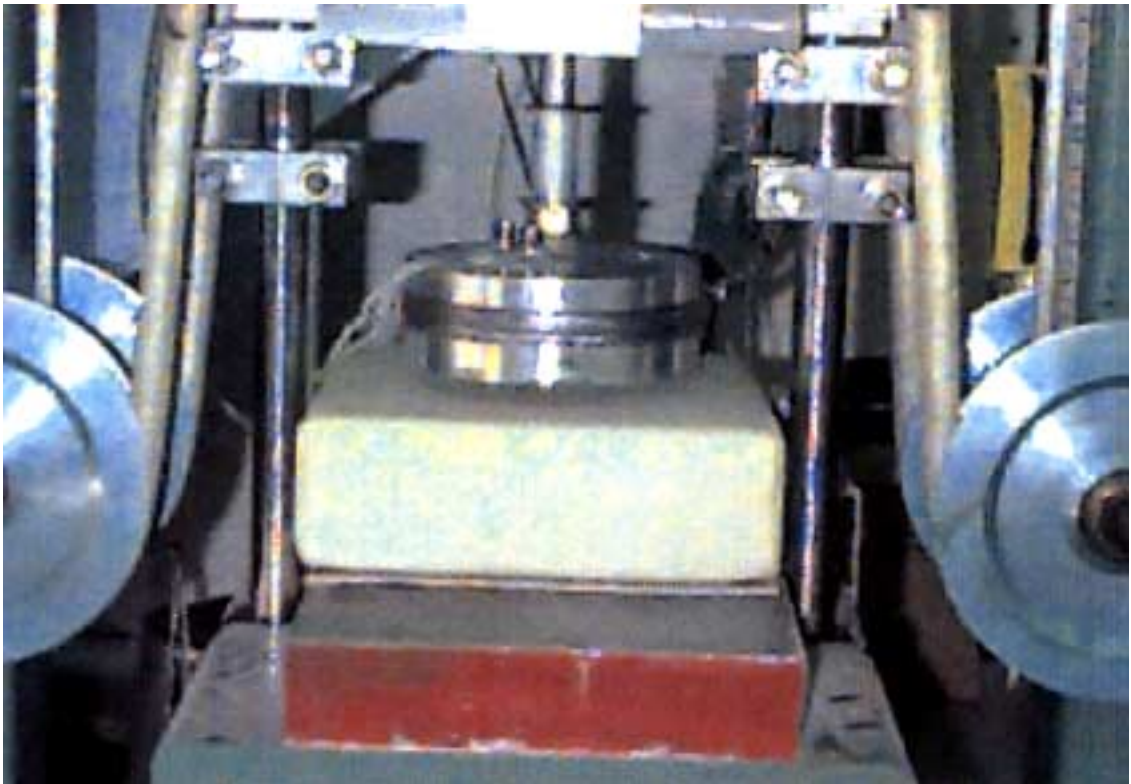
5.6.3.1 SEAT CUSHION MODELING USING MSC/DYTRAN

The seat cushion can be effectively modeled using the FOAM1 material model. The model assumes a crushable material where the Poisson's ratio is effectively zero. The yield behavior of the foam is determined by a stress-strain or crush-strain curve, typically obtained through a uni-axial compression test.

The crush-stress input data for the FOAM1 material model can be obtained by conducting a high velocity impact test (reference AGATE

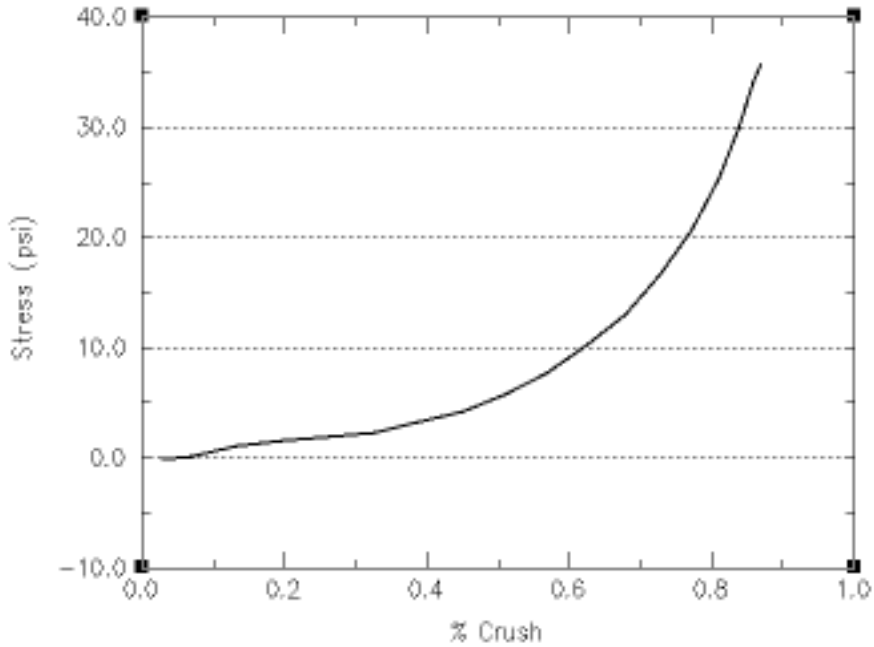
Report C-GEN-3432A-2 for test methodology, Figure 5-24). This impact test captures the dynamic response of the seat cushion material. In cases where the foam is not as sensitive to the rate of loading, a comparable static test (reference AGATE Report C-GEN-3432A-2 for test methodology) is sufficient to capture the response of the foam.

Figure 5-24 Foam Impact Test



As an example, an impact test was conducted on HR polyethelene foam of thickness and build-up that represents the actual seat cushion design. A leather fabric was sewn over the foam to represent the seat cover. A 51-lbm impactor was dropped on the foam sample at a velocity of 10 ft/s to obtain the bottom-out response of the seat cushion (Figure 5-25).

Figure 5-25 Stress-%Crush Foam Data



Since the MSC/DYTRAN FOAM1 material model does not incorporate hysteresis effects, only the loading function was used for the analysis. The corresponding FOAM1 input deck is shown in Figure 5-26

Figure 5-26 Example of FOAM1 material model

```

$ ----- Material foam id =11 HR 10/30
FOAM1      11 3.33e-6      75.00      6 CRUSH      +
+
+          DYNA  1.6    0.1
$ dynamic test data HR10 with 12"x12" leather cover
TABLED1    6      +
+,.0,.0,.067646,0.109,.132487,1.051,.187033,1.540,+
+,.261175,1.901,.324763,2.221,.387647,3.204,.449529,4.253,+
+,.510053,5.765,.568779,7.747,.625082,10.20,.678216,12.99,+
+,.727285,16.58,.771185,20.79,.808633,25.34,.838227,29.96,+
+,.858571,34.07,.868466,35.84,ENDT
  
```

5.7 APPLYING BOUNDARY CONDITIONS

Multi-body and finite element modeling requires the application of various boundary conditions. For dynamic seat modeling, typically only

nodal constraints and contact definitions are required. This may include the application of nodal displacement constraints (or SPCs) to the seat feet, for example, to represent attachment to the aircraft.

5.7.1 KINEMATIC CONSTRAINTS

Kinematic constraints consist of single (SPC) or multi-point constraints (MPC). In theory, kinematic constraints constitute the release or removal of a particular degree of freedom. For MPC's, the motion of a dependent degree of freedom is expressed as a linear combination of one or more independent degrees of freedom. In practical terms, they are used to tie a structure to the ground, to apply symmetric boundary conditions, to remove degrees of freedom that are not used in structural analysis, or to tie structures together. SPC's and MPC's in dynamic analysis are applied in the same manner as static finite element analysis. An in-depth discussion is not required.

A particularly effective constraint tool is the MSC/DYTRAN RCONN (equivalent to SPOTWELD in MADYMO) capability. RCONN is used to connect different parts of the mesh together without having to condense adjacent nodes. The desired effect is to simulate welded structures. RCONN represent a more sophisticated forms of MPC.

An example application of RCONN is shown in Figure 5-27. In Figure A, the gusset and the seat frame finite element mesh were modeled in separate groups. They are then tied together using the RCONN card to simulate a welded structure.

In MSC/DYTRAN, the RCONN input card structure is similar to the CONTACT card. The user needs to specify a set of slave nodes that will be tied to a master surface. The user will also need to define a monitoring distance, such that nodes with distance larger than the specified range will not be included in the connection. An example input deck is shown in Figure 5-28.

Figure 5-27 Rigid Connections

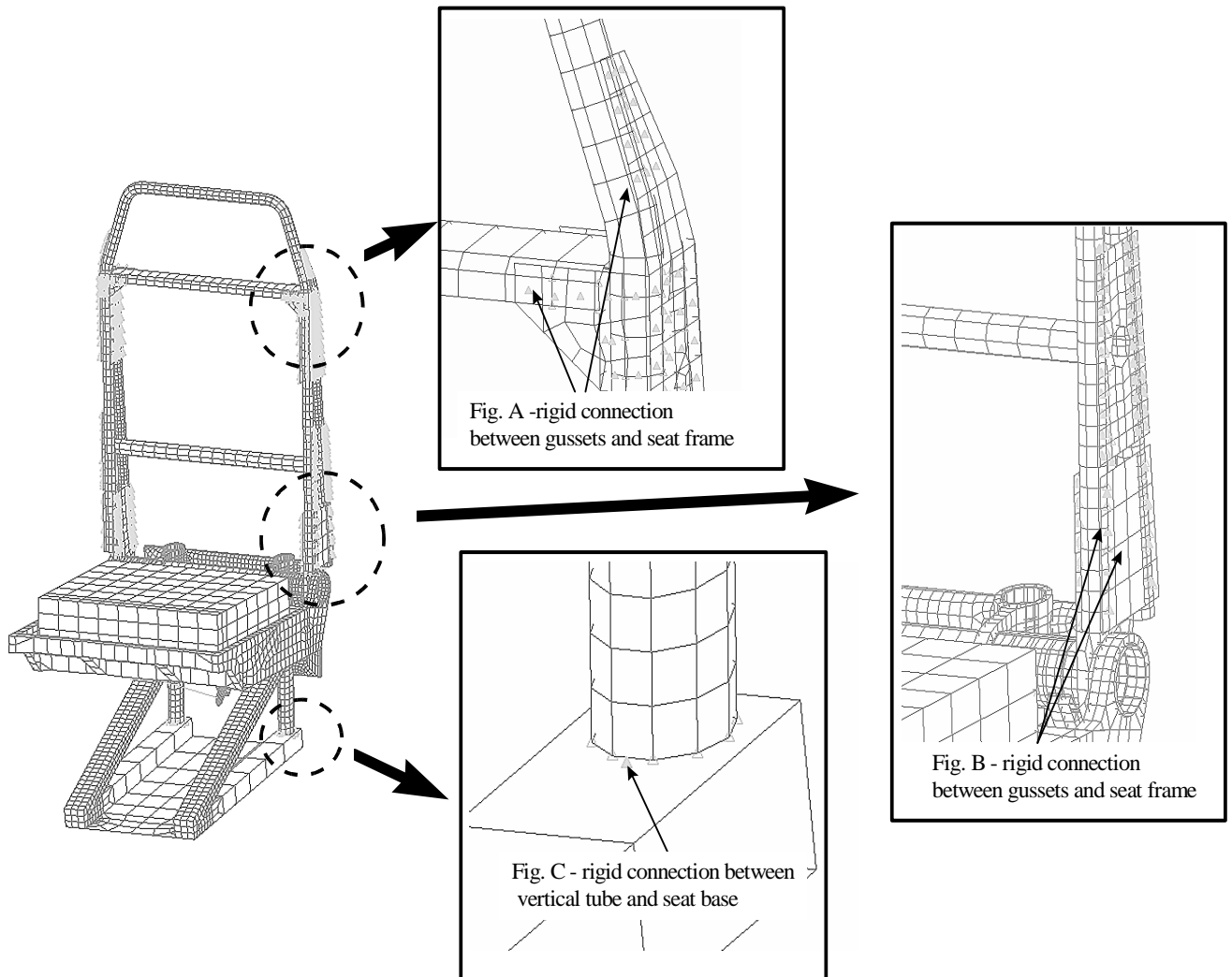


Figure 5-28 Example RCONN Input Deck

```

$----- web-tube weld
RCONN      26  GRID  SURF      1      2  NORMAL      +A000073
+A000073
+A000074      NODISTANCE  .020      +A000074
$
$ Slave contact surface for web-tube
$
SET1        1    1402    1405    1411    1414    1420    1423
$
$ Master contact surface for web-tube
$
SURFACE     2          SEG      2
CFACE       1          2      772    1
CFACE       2          2      773    1
CFACE       3          2      774    1
  
```

The slave nodes of the web (defined in SET1 entry) are connected rigidly to the master surface of the tube structure (defined in the SURFACE entry). The monitoring distance was set at 0.020 inches.

5.7.2 CONTACT DEFINITION

Contact definitions are required to evaluate the interactions between the seat and the occupant, between seat components, occupant self contact, occupant-to-occupant contact, and interactions with other objects in the aircraft interior. Without contact definitions, a simulation will allow entities to pass through the same space resulting in inaccurate simulation. Defining contact correctly requires understanding the assumptions of the contact algorithm used in the analysis code. Consult the software documentation for details on the limitations and assumptions. Although the algorithms vary for the different analysis codes, the input requirements are usually similar.

Generally, it is required to define master and slave contact entity sets (nodes, elements, rigid bodies). The rule of thumb is to assign the coarse mesh as the master surface and the finer mesh as the slave surface. If the mesh densities are similar, the slave surface should be the surface with the softer underlying material. Also, when selecting elements for contact analysis, it is better, in general, to use first-order elements for those parts of the model which will form a slave surface. This specifies which objects to include in contact interactions.

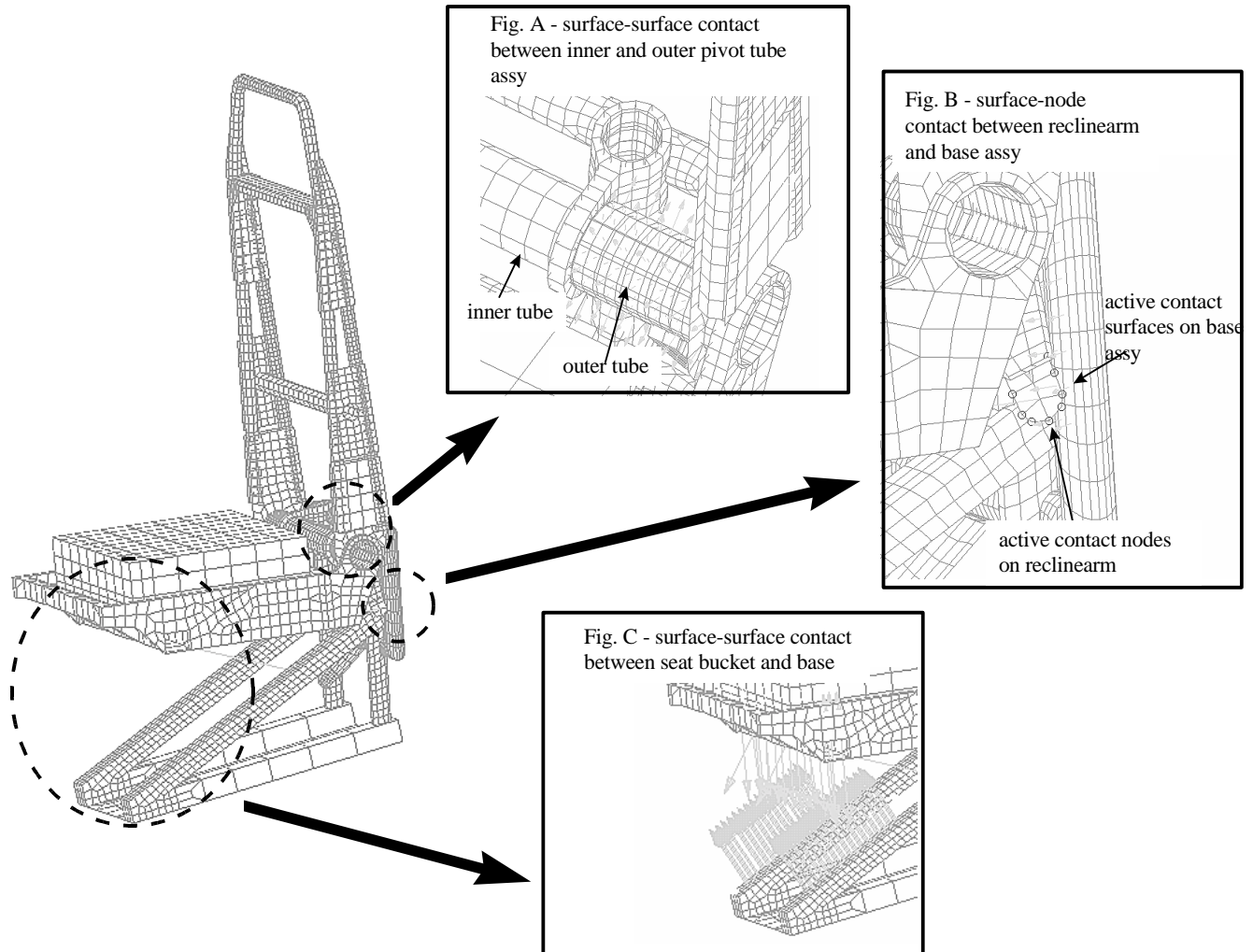
Static and dynamic friction coefficients are also required. Sometimes it is required to define contact surface direction, search radius, damping parameters, contact stiffness, contact start/stop time, or other parameters.

Initial penetration can be a problem in cases where finite element membrane mesh is applied to a multi-body occupant. Due to the discretization of the belt model, it is easy to have a few nodes that are just below the surface of the occupant "skin". Most contact algorithms calculate the contact force vector based on the restoration force required to move an intersecting slave node to the surface of the master entity. Without some method of compensation, the penetrated nodes will create enormous contact restoration forces resulting in numerical instability. To correct this, some codes can detect penetration of master and slave contact entities at the start of the simulation by turning on the penetration check option in the

contact definition card. The code will adjust the violating node(s) or offset the initial contact forces to equilibrium.

Surface-to-Surface and Surface-to-Nodes are the two most commonly used contact algorithms. Surface-to-Surface contact definitions are used where shell elements are anticipated to contact each other. Surface-to-Nodes contact is defined where the elements form a T-joint between surfaces. Examples of contact applications are shown in Figure 5-29.

Figure 5-29 Contact Applications



Since the contact search process is computationally expensive, it is recommended to minimize the entities included in the contact model. Predicting the kinematics of the simulation and estimating contact points can help to choose the appropriate contact entities. For example, simulating a pilot seat dynamic test, if it is expected that the occupant head will strike the instrument panel finite element mesh, choose a set of nodes in the expected contact area rather than the entire panel mesh. If necessary, the selected set can be adjusted after reviewing the results, and the simulation re-run.

5.7.2.1 DEFINING CONTACT WITHIN MSC/DYTRAN

An example MSC/DYTRAN surface-to-surface contact input deck is shown in Figure 5-30. The deck defines the contact interactions between the inner and outer pivot assembly shown in Figure A.

Figure 5-30 MSC/DYTRAN Surface Contact Definition

```

$ ----- Contact : inner to outer pivot assembly
CONTACT      299   SURF   SURF    73    74    .3    .3    .1+A000644
+A000644      V4   BOTH                    1.0                    +A000645
+A000645                                           0.1                    +A000646
+A000646                                           ON
$ Slave elements
SURFACE      73           SEG    73
CFACE        2163      73   5493    1
CFACE        2164      73   5494    1
.
.
CFACE        2171      73   9333    1
$ Master elements
SURFACE      74           SEG    74
CFACE        3252      74   5402    1
CFACE        3253      74   5403    1
.
.
CFACE        3271      74   5441    1
  
```

The contact specifies that the outer tube slave surface (defined by SURFACE 73) to be checked for contact with the inner tube master surface (defined by SURFACE 74). The outer tube is designated as the slave surface because it has a finer mesh in comparison to the inner tube. The value of the static and dynamic coefficient is 0.3. The contact employs a Version 4 algorithm (V4 input), which simultaneously tracks multiple contacts per slave node. By specifying BOTH in the contact card, slave nodes are checked for penetration on both sides of the master element regardless of the direction of the normal vector on the master surface.

5.7.2.2 DEFINING CONTACT WITHIN MADYMO

Contact in MADYMO can be defined between multi-body elements, and also between finite element models and multi-body elements. An elastic contact algorithm is used for multi-body contacts, where the contact characteristics are user defined and the resultant contact force is a function of penetration depth. A kinematic contact algorithm is used for finite element to multi-body contact problems, and the contact force is calculated based on the relative velocity of the node and contact surface. In most cases, contact characteristics are obtained from component test and specified in terms of force versus deflection. Contact is defined in MADYMO via the CONTACT INTERACTIONS card.

An example application of multi-body contact and input deck is shown in Figure 5-31 and Figure 5-32. The occupant's head (system 1, body 6) and the glareshield (system 6, body 2) are modeled as ellipsoids. The

ELLIPSOID-ELLIPSOID card is used, and the stiffness characteristic of the glareshield is defined in the FUNCTIONS card.

Figure 5-31 MADYMO Multi-Body Contact

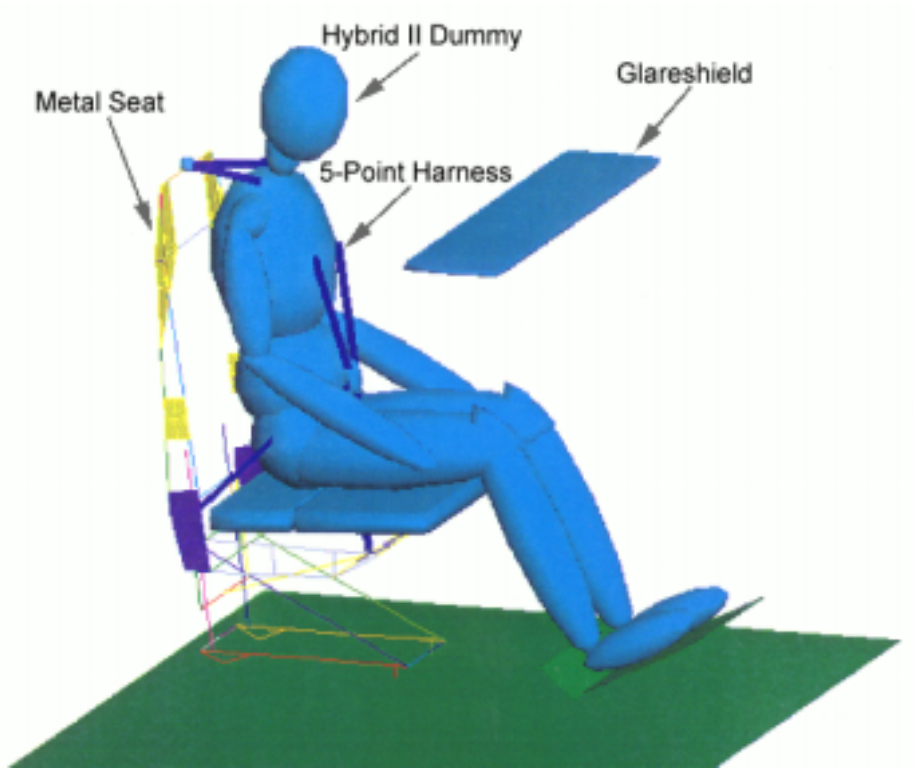


Figure 5-32 Multi-body Contact Definition

```
CONTACT INTERACTIONS
FUNCTIONS
ELLIPSOID-ELLIPSOID
! contact between glareshield and dummy's head
 6   2   1   6   4  -5   0  -1e+06  0.000000  +
      0.000000  0.300000  1   0   0
END ELLIPSOID-ELLIPSOID
FUNCTIONS
! glareshield load-deflection curve
 27
 0.0000  0.0000
 0.001  163.8842
 0.0030  326.5717
  .
  .
 0.0267  1873.9352
END FUNCTIONS
END CONTACT INTERACTIONS
```

5.8 LOAD APPLICATION

There are two methods to simulate crash loads.

Method One: Prescribe the initial velocity (or velocity prior to impact) to the seat and occupant, and apply deceleration to the sled. This method simulates the physical impact event, as experienced by the occupant.

Method Two: Apply the acceleration field to the occupant while maintaining the sled/seat as a stationary frame of reference. This method is an approximation of the impact event, because it assumes that the acceleration measured by the sled is the same as the occupant. The crash loads are applied in reverse of the actual physical event i.e. by applying an acceleration pulse to the occupant instead of a deceleration pulse to the seat and allowing the occupant to decelerate on its own. This method is acceptable only when the inertial effects of the seat are negligible in the direction of the applied load. The advantage of applying the acceleration field to the occupant is that it allows for the simulation of the 1 G pre-load, which is critical in predicting spine loads. Using this method, the impact acceleration can be offset by a certain amount of time to allow for the occupant to sink into the seat cushion.

5.8.1 LOAD APPLICATION FOR 60 DEGREES PITCH TEST

The illustration presented here is based on the second method described above. Two sets of load are required.

1. A 1 G gravity load in the negative Z direction (down) applied to the seat and occupant, and
2. The crash load simulating the 60 degrees pitch condition.

The crash acceleration profile used in the simulation can be in the form of an idealized triangular pulse per 23.562 or from actual test data.

5.8.1.1 EXAMPLE: LOAD APPLICATION WITH MSC/DYTRAN

Method 2 is utilized in this example. An acceleration field is applied to the occupant while maintaining the sled/seat as a stationary frame of reference. The 1 G gravity load is applied to the occupant via CARD A3 in the *ain data deck (Figure 5-33). The crash pulse is applied to the ATB occupant by means of the MSC/DYTRAN ATBACC and TLOAD card (Figure 5-34). The 60° vector is defined in the ATBACC card by using a load factor of (-0.5, 0.0, 0.866) in the (X,Y,Z) direction consistent with the direction of the occupant coordinate system. The crash pulse has an offset of 150 milliseconds from time zero to allow for adequate 1-G cushion pre-loading (Figure 5-35).

Figure 5-33 ATB 1 G Load Application Pitch Test

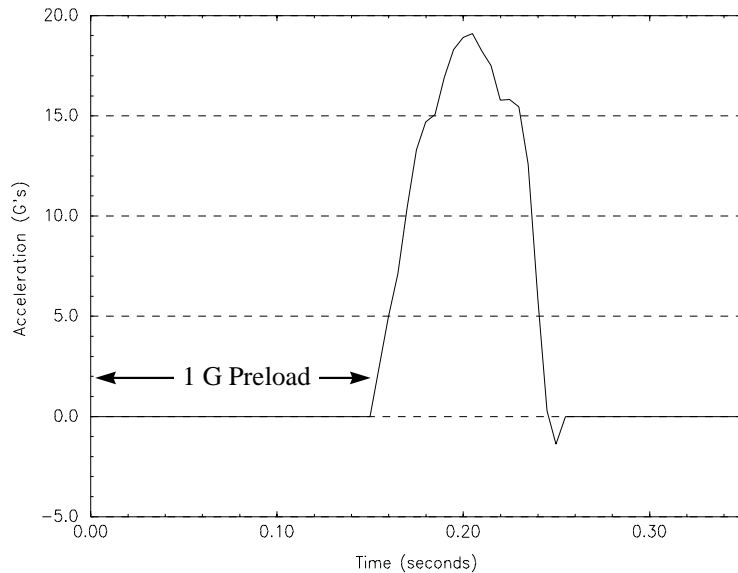
SITTING HYBRID II DUMMY (50%) GENERATED WITH GEBOD					CARD A1B
AGATE SLED TEST					CARD A1
IN. LB.SEC.	0.0	0.0	-386.088	386.088	CARD A3

Figure 5-34 MSC/DYTRAN Load Application Pitch Test

```

$Crash Pulse
ATBACC,201,,386.04,-.5,0.0,-.866,,,+
+,LT,MT,UT,N,H,RUL,RLL,RF,+
+,LUL,LLL,LF,RUA,RLA,LUA,LLA
$
TLOAD1,13,201,,,1000
TABLED1,1000,,,,,,,,,+
$ ACCELERATION WITH 0.15 SEC 1 G LOAD
+,0.0,0.0,0.150,0.0,0.16,4.92633,0.165,7.11431,+
+,0.17,10.4175,0.175,13.2985,0.18,14.6757,0.185,15.0433,+
+,0.19,16.9036,0.195,18.296,0.20,18.8951,0.205,19.0857,+
+,0.21,18.2143,0.215,17.4943,0.22,15.7737,0.225,15.8078,+
+,0.23,15.434,0.235,12.5829,0.24,5.92312,0.245,0.26516,+
+,0.25,-1.39478,0.255,0.0,0.35,0.0
$
  
```

Figure 5-35 Test 1 Applied Loads



5.8.2 LOAD APPLICATION FOR 10 DEGREES YAW TEST

The illustration presented here is based on Method 1. Two sets of load are required.

1. A 1 G gravity load in the negative Z direction (down) applied to the seat and occupant, and

2. The crash load simulating the 10 degrees yaw condition.

The crash acceleration profile used in the simulation can be in the form of an idealized triangular pulse per 23.562 or from actual test data.

5.8.2.1 EXAMPLE: LOAD APPLICATION WITH MSC/DYTRAN

A 1-G gravity load is applied in the negative Z-direction. The 1-G load is applied to the seat via the MSC/DYTRAN TLOAD1 and GRAV card. The crash scenario is simulated by prescribing an initial velocity prior to impact to all elements in the model, and applying a deceleration field to the sled.

The ATB initial velocity is prescribed in the ATB input deck using the G2 card. All other MSC/DYTRAN elements receive the initial velocity definition through the TICGP card. Both ATB and MSC/DYTRAN initial velocities are defined at a vector of 10^0 from the horizontal plane to simulate the yaw condition. The sled is decelerated by prescribing a velocity profile (Figure 5-36) to all of the elements of the sled using the TLOAD1 and FORCE cards.

Figure 5-36 Test 2 Applied Loads

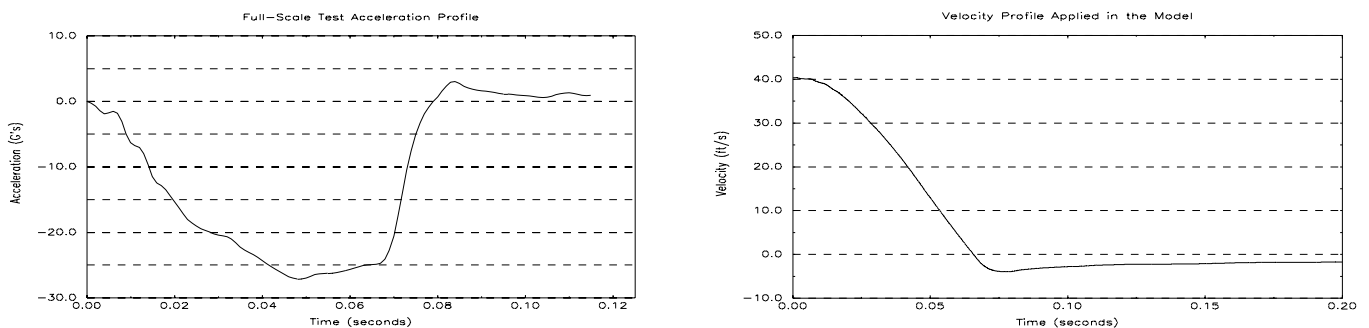


Figure 5-37 MSC/DYTRAN Load Application Yaw Test

```

$ATB Input Deck: Initial velocity definition
  -117.523      0.00  -27.9607   -476.9      84.07      0.00      CARD G2
$
$MSC-DYTRAN Input Deck: Initial velocity & prescribed motion definition
$ ----- GRAVITATION -----
$1 G Load applied to the seat
TLOAD1      13      444      0
GRAV      444      32.17      0      0      -1.0
$ ----- Initial Velocity BC initial velocity entire model -----
TICGP      13      200      XVEL  -476.9      YVEL  84.09
$
SET1      200      1      THRU  2548      2550      THRU  4413      4421+A000652
+A000652  THRU  4509      4515      THRU  8026      8139      THRU  8538+A000653
+A000653  9000      THRU  9135      12020      THRU  13102      14500      14501+A000654
$ ===== PRESCRIBED SLED MOTION =====
$ Apply prescribed velocity profile to rigid elements that represents the sled
TLOAD1      13      294      2      90
FORCE      294      12020      0      1  -.9848  .1736      0
FORCE      294      12024      0      1  -.9848  .1736      0
.
.
FORCE      294      44479      0      1  -.9848  .1736      0
$ Pulse from crash test
$ ----- TABLE 90: velocity_table -----
TABLED1      90
+A000658      0      484.3      .004      483.3      .005      481.6      .007      480.8+A000658
+A000659      .009      480      .0094      478.7      .0098      477.1      .01      476.3+A000660
+A000660      .0105      474.5      .011      473      .0115      471.8      .0125      469.7+A000661
+A000661      .0135      467.7      .0145      464.4      .015      461.7      .02      435.5+A000662
+A000662      .03      363.7      .04      278.4      .05      177.3      .055      126.3+A000663
+A000663      .06      75.9      .065      27.3      .067      8.3      .0678      .76+A000664
+A000664      .15      0      ENDT

```

The data deck in Figure 5-37 shows the finite element nodes (specified by SET1 and TICGP card) and ATB (last three entries of the G2 card) has initial velocities of (-476.9,84.09,0.0) in/s. This translates to a resultant impact velocity of 484.2 in/s (40.35 ft/s). The TLOAD1 card then prescribes a velocity change for the sled (elements 12020 through 44479 defined in the FORCE card) as specified in TABLE1 card profile.

5.9 FLOOR DEFORMATION

The specific method for simulating floor deformation is code dependent. In MSC/DYTRAN, the 10^0 pitch and roll seat legs floor deformation can be simulated by prescribing a time dependent velocity profile to the nodes corresponding to the location of the seat feet attachments. Integration of the velocity profile will yield the required displacement of the seat legs nodes, which will then create a pre-stress on the seat.

In MADYMO, floor deformation can be accomplished by enforcing a prescribed displacement. In either case, enforced motion in transient analysis can result in numerical instability if it is incorrectly executed.

5.9.1 EXAMPLE FLOOR DEFORMATION SIMULATION USING MADYMO

Floor deformation can be modeled in MADYMO by using null systems and attaching it to the seat feet fittings (a null system is used to model a system of a body with known motion relative to inertial space). Each null system, attached to the inboard or outboard seat leg, is prescribed a 10^0 pitch and 10^0 roll motions corresponding to actual test requirements. Typically, floor deformation simulation has to be performed for a minimum duration of 200 milliseconds to avoid numerical instability. An example input deck illustrating floor deformation using null systems is shown in Figure 5-38.

Figure 5-38 Floor Deformation Using MADYMO

```

NULL SYSTEM
INBOARD FWD LEG PITCH DOWN
MOTION
POSITION
0.0 0.05 -0.2286 -.05
0.01 0.055 -0.2286 -.10
0.35 0.055 -0.2286 -.10
END POSITION
END NULL SYSTEM
NULL SYSTEM
OUTBOARD FWD LEG ROLL OUT
MOTION
POSITION
0.0 0.05 -0.4445 -.05
0.01 0.05 -0.47 -.04
0.35 0.05 -0.47 -.04
END POSITION
END NULL SYSTEM
NULL SYSTEM
OUTBOARD AFT LEG ROLL OUT
MOTION
POSITION
0.0 0.05 -0.4445 -.05
0.01 0.05 -0.47 -.04
0.35 0.05 -0.47 -.04
END POSITION
END NULL SYSTEM
! constrain inboard fwd leg nodes to null system 1- pitch down
  NUMBER          6
  NULL SYSTEM     1
  DOF             ALL
  SET             82
! constrain outboard fwd and aft leg nodes to null system 2 roll out
  NUMBER          7
  NULL SYSTEM     3
  DOF             ALL
  SET             62
  
```

5.9.2 EXAMPLE FLOOR DEFORMATION SIMULATION USING MSC/DYTRAN

The floor deformation requirements can be simulated using a combination of the TLOAD1 and FORCE3 card. An example input deck is shown in Figure 5-39.

Figure 5-39 Floor Deformation Using MSC/DYTRAN

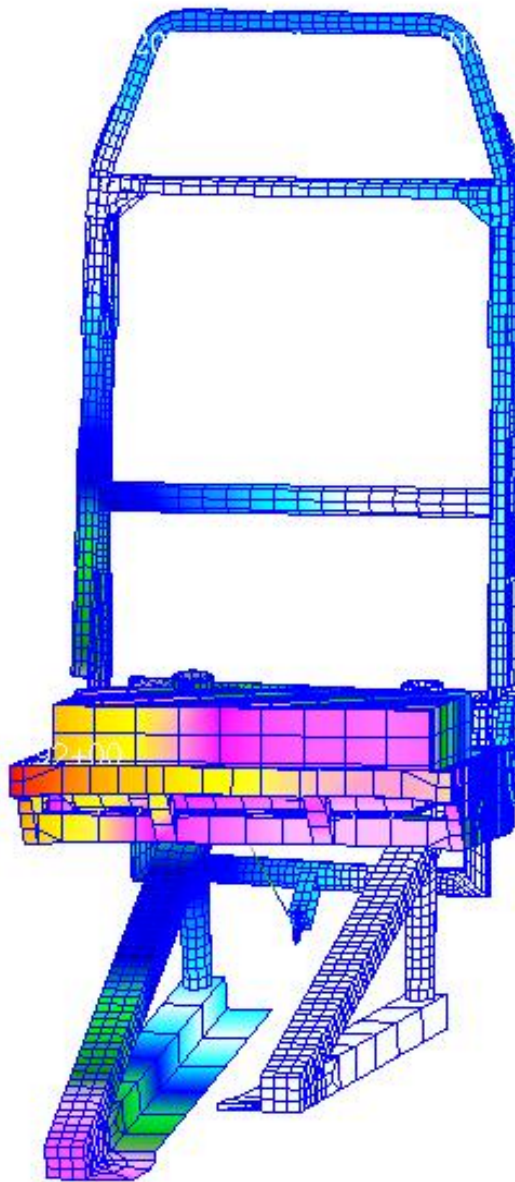
```

=====
$ ===== FLOOR DEFORMATION: 10 DEGREES PITCH & ROLL =====
$=====
$
$ Inboard leg rotation coord system
CORD1C      1      368      346      54
$ Outboard leg rotation coord system
CORD1C      2      346      54      80
$
$ 10 deg. Pitch down on inboard seat leg
TLOAD1      1      995      2      995
FORCE3      995      80      1  5.8178      1.      0.
FORCE3      995      323      1  5.8178      1.      0.
FORCE3      995      366      1  5.8178      1.      0.
.
.
.
FORCE3      995      104      1  5.8178      1.      0.
TABLED1      995
+      0.0      0.0      0.02      1.0      0.03      1.0      0.05      0.0+
+      1.0      0.0      ENDT
$
$ 10 deg. Outbd roll outboard seat leg
TLOAD1      1      996      2      996
FORCE3      996      332      2  5.8178      -1.
FORCE3      996      344      2  5.8178      -1.
.
.
.
FORCE3      996      78      2  5.8178      -1.
TABLED1      996
+      0.0      0.0      0.02      1.0      0.03      1.0      0.05      0.0+
+      1.0      0.0      ENDT

```

A coordinate system is specified for each inboard and outboard seat rail, by means of the COOR1C card, to define a rotation axis for the seat leg. The TLOAD1 card then specifies the nodes, defined in the FORCE3 cards, to rotate in accordance with the COOR1C rotation definition and the velocity change function in TABLE1 card. Figure 5-40 shows a seat with floor deformation simulation using the technique described above.

Figure 5-40 MSC/DYTRAN Pitch and Roll Simulation



6. GENERAL DISCLAIMER

This document serves as a guideline in the application of computer models for the design and certification of seat/restraint systems. The information presented reflects the state of the computer

simulation technology at the time this document was developed. Specific modeling techniques may in fact become obsolete as technology progresses. In addition, there are other methods of modeling seating systems that are equally valid but are not specifically covered in this report.

Although this document provides detailed guideline to the certification of seat designs, it does not guarantee certification. The user must still coordinate and gain approval with the FAA on the specifics related to the product's certification.

To the extent permitted under applicable law, the user accepts responsibility for any liability stemming from the application of this document, and agrees to hold Cessna Aircraft Company and any member of AGATE harmless from any claims of liability arising from the user's commercial use of the information produced in this document.

7. ACKNOWLEDGEMENTS

Special appreciation to the following members of the AGATE Advanced Crashworthiness Group who provided valuable guidance and criticism in the development of this document.

Mr. Pat Mullen - FAA Small Airplane Directorate

Mr. Steve Soltis - FAA National Resource Specialist

Dr. Steve Hooper - AGATE Systems Engineer

Dr. Joseph Pelletteire - Wright Patterson Air Force Research Labs

Dr. Karen Lyle - NASA Langley

Dr. Ken Lou - Simula Technologies Inc.

Mr. Todd Hurley - Simula Safety Systems Inc.

Ms. Marilyn Henderson - National Institute Aviation Research, Wichita
State University

Mr. Thomas Hermann - Raytheon Aircraft Company

Ms. Sonja Englert - ModWorks

Mr. Paul Yaklin - Cessna Aircraft Company

Mr. Todd Bevan - The Lancair Company