The Current Status and Future Development of the Interface between MSC.Patran and MSC.Marc

Tim Kuhlmann, Ph.D.
MSC Software Mechanical Solutions Division
2975 Redhill Ave.
Costa Mesa, CA 92626
Email: tim.kuhlmann@mscsoftware.com

ABSTRACT

The new combination of MSC.Patran's graphical user interface for comprehensive automated meshing and finite element (FE) model manipulation and MSC.Marc's extensive suite of nonlinear material modeling capabilities, robust element and solver technology, and powerful but user-friendly contact features should soon[1] be the most powerful, and still easy to use, software combination available anywhere for performing nonlinear structural and thermal finite element analysis (FEA). MSC.Marc is one of the world's leading finite element analysis programs for solving complex, highly non-linear, analysis problems including geometric, material, and boundary (particularly contact) nonlinearities such as occur in large-deflection, large-strain contact problems, elastomeric component behavior and manufacturing simulations. MSC.Patran is a finite element pre- and post-processing software package that was developed to create finite element models from geometry, which is either imported from computer aided design (CAD) software packages, or built completely within MSC.Patran. MSC.Patran provides tools for creating the mesh, material models, loads and boundry conditions (LBC's), and inputting the analysis parameters required to completely describe the problem in the input format required by the analysis code (in this case, MSC.Marc). The combination of MSC.Patran's ability to: a) import and export data to a wide variety of CAD and other FE analysis programs, b) repair non-congruent geometry, perform model manipulation, and produce a mesh on even the most complex geometries, and c) describe complex analysis load, constraint and contact histories; together with MSC.Marc's mature, feature-rich, powerful nonlinear analysis capabilities make this combination a robust, user-friendly tool for performing nonlinear finite element modeling and analysis. The interface between MSC.Patran and MSC.Marc, called the MSC.Marc analysis preference for MSC.Patran, is being extended to support almost all of the analysis capabilities of MSC.Marc. The MSC.Marc preference in MSC.Patran 2000 (versions 9.0) and lower had not kept up with changes in the latest releases of the MSC.Marc solver. In addition, there were code defects (i.e. bugs), documentation errors and other deficiencies that made it difficult to build and run MSC.Marc models from MSC.Patran. The commonly-used procedure was to build the mesh in MSC.Patran and then build the rest of the model in Mentat (MSC.Marc's own GUI pre/post-processor). Starting with MSC.Patran 2000 R2 (version 9.5) it should be possible to build the entire model in MSC.Patran with all contact and LBC, almost all element types, many material types, and the most common analysis procedures supported. Version 2000 R2 is the first one to include an MSC.Marc input deck reader. MSC.Patran version 2001 will be the first version to fully support all LBC, material, and element formulation options; as well as nearly all of the analysis procedures and capabilities, such as domain decomposition and multi-stepping. Multi-stepping is the ability to set up complex contact, load and constraint histories, including changing analysis procedures, without using restarts. There are a few MSC.Marc capabilities, such as fully coupled thermal-structural analysis and some of the latest features recently added to MSC.Marc, that will not be supported until the MSC.Patran release following version 2001. This paper provides the details of the revised interface along with the approximate time frame when the new capabilities will be available. It also answers frequently asked questions regarding switching to the MSC.Patran/MSC.Marc combination from MSC Software's current nonlinear analysis product, MSC.Patran Advanced FEA.

[1] Note that since nearly all aspects of the MSC.Marc preference mentioned in this paper are currently under active development, all time estimates are best guesses according to current plans. The actual implementation may vary.

SUMMARY

The graphical user interface (GUI) connecting MSC.Patran and MSC.Marc (Figure 1), called the MSC.Marc analysis preference for MSC.Patran, is being extended to support almost all of the analysis capabilities of MSC.Marc. The combination of MSC.Marc's powerful, fully-integrated and comprehensive non-linear analysis procedures with Patran's feature-rich, general purpose modeling capabilities should provide one of the best software packages available for performing nonlinear structural static and dynamic, thermal, and fully coupled thermal-structural simulations. This package will be particularly advantageous for simulating complex problems such as manufacturing operations involving multiple bodies and closely controlled loading, contact and constraint histories. The combination should easily handle analyses involving combinations of analysis procedures such as stress-stiffened modal or buckling analysis; and thermal, or some other influencing variable, results being mapped on to affect the structural behavior.

Analysis Capabilities and Solution Procedures: The MSC.Patran 2001 release (version 10) is going to include major new capabilities in the MSC.Marc analysis preference (Figure 1). These new capabilities will include new analysis procedures, multi-stepping for structural and thermal analysis (Figure 2) and, in a release beyond MSC.Patran 2001 (referred to in this paper as "a future release," and will come out sometime in the year 2001), will include support for coupled thermal-structural analysis. The analysis procedures to be supported include linear and nonlinear static, normal modes, buckling, transient dynamic (including linear and nonlinear, modal and direct), frequency response, spectrum response, and creep for structural; steady state and transient for thermal; and linear and nonlinear static, transient dynamic and creep for coupled thermal-structural. Many of the new capabilities in version 2001 are going to be included under the translation parameters form (Figure 3.). These new capabilities include direct text input, the conversion of Patran groups to Marc sets, element activation and deactivation between steps, both global and local adaptive re-meshing, and (automated or manual) domain decomposition control supported through the GUI. In the future release mentioned earlier, the Patran analysis manager, which can be used to control the submission of a single job, multiple jobs or a single job broken up into multiple jobs, and the capability to stack analysis jobs will also be supported. It will be possible to include user subroutines in the job submission process in version 2001, however, the application of specific LBC, material property and element formulation subroutines to specified parts of the model will not be supported until the next release after version 2001.

Loads, Boundary Conditions and Contact: The version 2001 release will include all of the available structural and thermal loads and boundary conditions supported by MSC Marc, including multi-body contact parameter specification and rigid body velocity, displacement and force translation and rotation control (only velocity control is supported in version 2000 R2). The contact analysis capabilities supported by the GUI in version 2001 will include nearly all of the contact parameters and options for discrete body (element) application regions, and be extended in the release after 2001 to include rigid surface geometry, rigid line geometry and rigid body motion control.

Material Properties and Element Formulations: The element property capabilities supported in version 2001 (Figure 4) have been extended significantly to include all MSC.Marc structural and thermal analysis elements; and in the future release mentioned will include elements for fully coupled thermal-structural properties and user subroutines. There are also significant new capabilities supported for re-bars, where both two-dimensional and three-dimensional layer tools will be supported. In the version 2001 release the beam library capability of MSC.Patran will be expanded to include the MSC.Marc preference. Also, the capability to perform experimental data fitting (currently only available through the GUI in Mentat) for all modes of test data, including visco-elastic relaxation, using MSC.Patran fields to import the raw data will be implemented in MSC.Patran. In version 2001 all of the Marc material models for linear and hyper-elastic (Mooney-Rivlin, Arruda-Boyce, Gent, Ogden and Foam), elastic-plastic, rigid and perfectly plastic, creep, failure, damping and visco-elastic analysis are supported (Figure 5). Those not supported include powder and soils. Support for material property user-subroutines is planned for the next release beyond 2001.

Results Post-Processing: The results post processing capabilities of Patran have also been extended to support the MSC.Marc 2000 results format, including support for all of the new post codes of MSC.Marc 2000. The MSC.Patran 2001 release will also include direct results access, which means that the results can be attached to the Patran database without having to be imported. In a future release the capability to do direct model access, which provides the ability to post-process using different meshes in each analysis increment (which is required to support

adaptive meshing and re-meshing) will also be included. Also supported in the future release will be the ability to post-process rigid body motion.

BACKGROUND - MSC.Marc

MSC.Marc (Marc) was the first commercial, and is still one of the world's leading finite element analysis programs (sometimes referred to as a finite element "solver"), developed specifically to solve highly nonlinear structural and thermal analysis problems. Types of non-linearity include large deformation, finite strain, material and boundry (contact). MSC.Marc provides robust user-friendly contact and has an extensive suite of material models and element formulations suited for non-linear analysis. The ability to analyze highly non-linear structural and thermal behaviors makes Marc a valuable tool for solving complex manufacturing, component stress analysis, and other types of simulations.

BACKGROUND - MSC.Patran

MSC.Patran (Patran) is a finite element pre- and post-processing software package that was developed to create finite element models using geometry that is either imported from computer aided design (CAD) software packages or built completely within MSC.Patran. MSC.Patran provides tools for creating the mesh, material models, loads and boundary conditions, and other analysis parameters required to completely describe the problem in the input format required by the analysis code (in this case MSC.Marc). MSC.Patran is a feature rich general-purpose finite element modeler which includes one-, two- and three-dimensional solid modeling and meshing capabilities. It has interfaces to all major CAD programs and many other analysis solvers, both implicit and explicit for structural, thermal and CFD analysis.

CONSIDERATIONS WHEN SWITCHING FROM ADVANCED FEA TO MSC.Marc

When MSC Software and the Marc Analysis Research Corporation merged in 1999, MSC's FEA product for solving highly non-linear problems was MSC.Patran Advanced FEA (AFEA). Since the merger, MSC has begun the process of switching AFEA users over to the Patran/Marc combination. The status of the MSC.Marc preference is of major importance to this process. For Advanced FEA users who are switching to MSC.Marc here are the answers to a few frequently asked questions:

Q: Will the MSC.Patran/MSC.Marc combination have all of the capabilities of Advanced FEA?
A: Current plans are that it will soon have everything and a lot more. The only item not supported to the same extent as it is in AFEA is random vibration analysis, although, it is possible to do this in MSC.Marc. In addition to having all of the capabilities of AFEA, MSC.Patran and MSC.Marc will have the following capabilities AFEA lacks: user subroutines, full 3D deformable body contact, higher order element contact, both global and local adaptive re-meshing, super-elements and domain decomposition, hour glass control for reduced integration elements, generalized plane strain, access to the input deck, support for the MSC.Patran beam library, direct results and direct model access for results post-processing, fully coupled thermal-structural and multi-physics analysis, and many more features. Anything you can do in AFEA, and more, can be done just as easily in MSC.Patran/MSC.Marc (when the preference supports it).
Q: Will my AFEA models run directly in MSC.Marc?
A: Models that involve standard shell and solid elements, linear elastic material models, displacements, forces, and pressures ONLY generally allow for the MSC.Patran database preference to be changed without lose of model information, and the model to run directly in MSC.Marc. Models containing more advanced features such as nonlinear material models, gap and beam elements, multi-stepping, mpc's and more complex capabilities that vary from one solver to the next in their implementation will likely require those advanced features to be re-created (or at least checked) after the database preference has been changed. As time and development resources allow, MSC will try to address this question by modifying the MSC.Patran database upgrade script (which updates the database from one version to the next) to "hardwire" some of the model entity conversions that aren't handled under general preference switching.

Q: When will all of these new capabilities be available in the MSC.Marc preference?
A: The MSC.Marc preference in MSC.Patran 2000 (versions 9.0) and lower had not kept up with changes in the latest releases of the MSC.Marc solver. In addition, there were code defects (i.e. bugs), documentation errors and

other deficiencies that made it difficult to build and run MSC.Marc models from MSC.Patran. The commonly-used procedure was generally to build the mesh in MSC.Patran and then build the rest of the model in Mentat (MSC.Marc's own GUI pre/post-processor). Starting with MSC.Patran 2000 R2 (version 9.5) it should be possible to build the entire model in MSC.Patran with all LBC, most material types, and almost all element types supported.

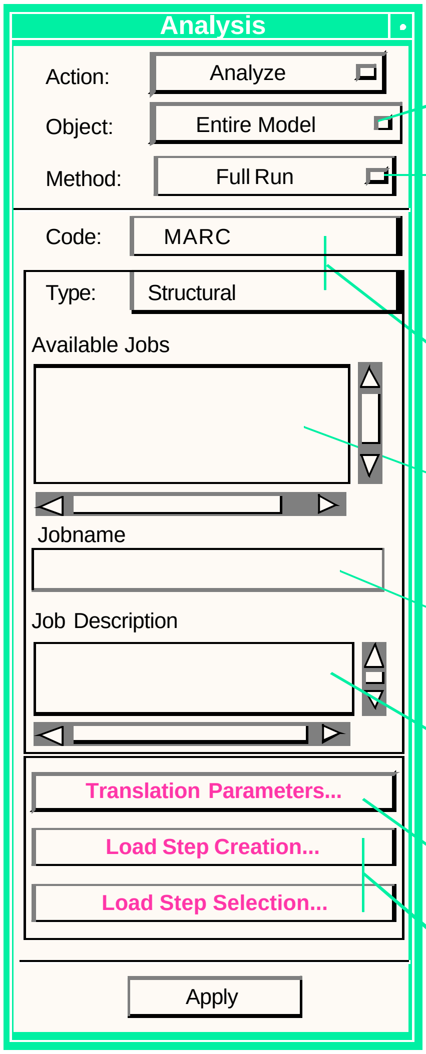
The major AFEA capability missing in the version 2000 R2 MSC.Patran Marc preference will be multi-stepping, which is tentatively (again it should be emphasized that these estimates are best guesses according to current plans, the actual implementation could vary) scheduled for MSC.Patran 2001 (version 10). And even in version 2000 R2 (version 9.5) you can do anything you can with multi-stepping by using the RESTART option (which is fully supported in version 2000 R2). The only thing to remember about multi-stepping in MSC.Marc using restarts is that the loads default to incremental, not total, values. So if you wanted to move the end of a cantilever beam down 1 unit in step 1, and then over 1 unit in step 2 you would have to apply a displacement of -1.0 in the vertical direction in step 1, and in step 2 apply a vertical displacement of 0.0 (if you fail to include this MSC.Marc will remove the constraint from step 1 and the tip will start step 2 from the un-deformed position) and a horizontal displacement of 1.0. Thus, you should be able to successfully build and run most models in version 2000 R2, and most multi-step models in version 2001. Some capabilities, such as fully coupled thermal-structural analysis will be available immediately with MSC.Marc by editing the input deck (or using direct text input when it is available in version 2001), but it is not planned that the MSC.Patran GUI fully support them until sometime after version 2001. And of course all of the MSC.Marc capabilities are supported today through Mentat, so they are available immediately using the Mentat GUI.

Q: When is the best time to switch from AFEA to MSC.Marc?
A: The answer to this question depends on several factors:

a) When will your AFEA license expire?
b) Do you require multi-stepping capabilities (or are you willing to use restarts in place of multi-stepping)?
c) When will the multi-stepping capabilities actually be available?
d) Will you benefit from the new capabilities available in MSC.Marc?
e) Do you have a significant number of AFEA models that will need to be converted?
f) Are you open to learning and using Mentat until the MSC.Patran preference supports what you need?

In general, if your AFEA license is not expiring and forcing you to change sooner, and if the current capabilities in AFEA are all that you need, the most trouble free course of action will be to wait until MSC.Patran supports the multi-stepping capabilities (tentatively version 2001), since that capability should change the way the analysis model is created. This will affect the session and journal files MSC.Patran builds and uses as backup. Until the multi-stepping capabilities are in, there will be compatibility issues of journal and session files from previous versions not working in the latest version of the preference.

Figure 1. Analysis Form



Indicates how much of the model is to be included in the analysis. This can be set to Entire Model or Current Group.

Indicates how far the analysis job will proceed. When set to Analysis Deck, the job stops once the translation is performed. For Check Run the analysis stops after the job is submitted and the work space has been allocated. For Full Run, the MARC analysis can go to completion.

Indicates the selected Analysis Code and Analysis Type. For MSC.Marc the planned options include structural, thermal, and coupled.

Lists all previously created MARC jobs. Selecting a jobname reloads the jobname and parameters previously used.

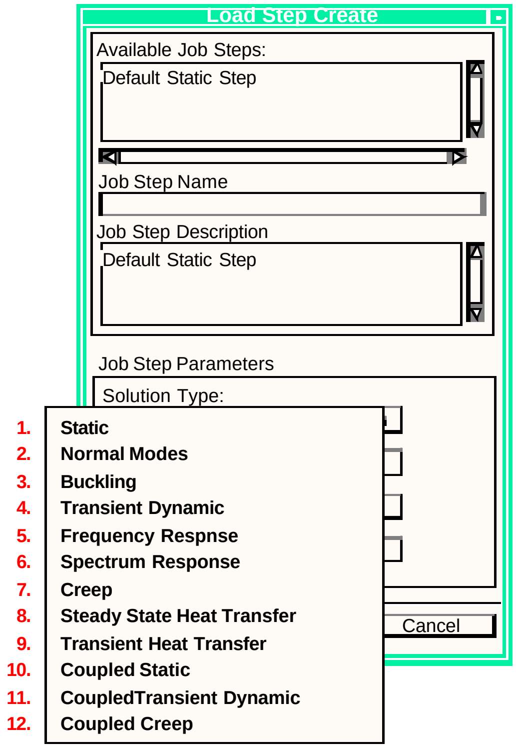
Defines the name to be used for all parameters, results cases, and files associated with this analysis.

Describes the analysis job. This generates the MARC TITLE option.

See Figure 2.

Complex analysis procedure historys, including contact, load and constraint changes, and even mesh changes, can be set up by describing the job as a series of analysis "steps."

Figure 2. Load Step Creation - Solution Types



Complex analyses can be described as a series of steps.

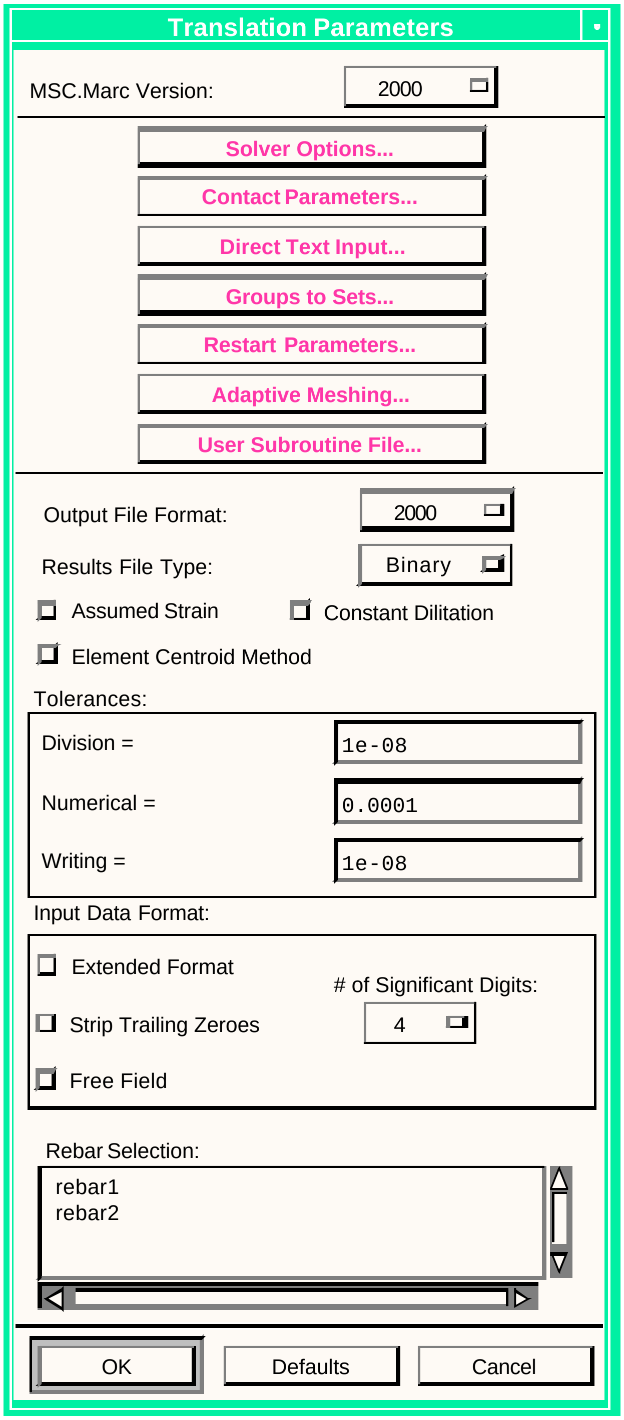
Within each step things that may change include:

- Analysis procedure
- Contact conditions
- Loading and constraints
- Mesh (elements may be activated and deactivated)
- Highly distorted areas can be sub-divided or completely re-meshed
- Output requests

Complex loading histories describing things like manufacturing simulations can be set up easily by describing a series of steps

Analysis inter-dependancies such as stress stiffening for buckling or modal eigenvalue analysis can be included easily by running one, or a series of, nonlinear static load steps before the eigenvalue analysis is performed.

Figure 3. Translation Parameters Form



Many of the characteristics of the analysis are controlled though the analysis parameters form.

Some of the MSC.Marc analysis features supported through the MSC.Patran graphical user interface (GUI) include:

- Multiple versions of Marc
- Equation optimizer and solver selection
- Contact body and parameter control through contact tables
- Model manipulation using node and element sets
- Restart job reading and writing
- Global and local adaptive re-meshing based on a variety of criteria
- User subroutines
- Input deck format and job parameter control

Any MSC.Marc feature not directly supported through the GUI can be easily accessed through the direct text input capability.

Figure 4. Supported Element Types

1D Elements (Structural)

Type	Option 1[8]	Option 2	Bar/2	Bar/3	Input Properties Form label[8]:
Elastic Beam	General Section	Standard Formulation	31		Beam w/General Section (31)-Bar/2
		Euler-Bernoulli	52		Euler Beam w/General Section (52)-Bar/2
		Euler-Bernoulli w/Shear	98		Euler Beam w/General Section (98)-Bar/2
	Arbitrary Section	Standard Formulation	31		Beam w/Arbitrary Section (31)-Bar/2
		Euler-Bernoulli	98		Euler Beam w/Arbitrary Section (98)-Bar/2
	Curved w/ Arbitrary Section		31		Curved Beam/Arbitrary Sect (31)-Bar/2
	Curved w/ General Section		31		Curved Beam/General Sect (31)-Bar/2
	Curved w/ Pipe Section		31		Curved Pipe (31)-Bar/2
	Pipe Section		31		Pipe (31)-Bar/2
Thin Walled Beam	Closed Section	Standard Formulation	14		Closed Section Beam (14)-Bar/2
		Linear Axial Strain	25		Closed Section Beam (25)-Bar/2
		Shell Stiffener	78	76	Closed Section Beam (78)-Bar/2, (76)-
	Open Section	Warping	79	77	Open Section Beam w/Warping (79)-Bar/2,
		Standard Formulation	13		Open Section Beam (13)-Bar/2
	Pipe Section	Standard Formulation	14		Pipe (14)-Bar/2
		Linear Axial Strain	25		Pipe (24)-Bar/2
		Shell Stiffener	78	76	Pipe (78)-Bar/2, (76)-Bar/3
Truss			9	64	Truss (9)-Bar/2, (64)-Bar/3
Spring			SPRING		Spring
Damper			SPRING		Damper
Gap	Fixed Direction		12		Fixed Directional Gap (12)-Bar/2
	True Distance		12		True Distance Gap (12)-Bar/2
	Friction w/Bending[2]		97		Gap/Friction Link w/Bending (97)-Bar/2/4
Cable	Initial Stress Input		51		Initial Stress Cable (51)-Bar/2
	Length Input		51		Iniitial Length Cable (51)-Bar/2
Axisym Shell	Homogenous or Laminate	Standard Formulation	1	89	Axisymmetric Shell (1)-Bar/2, (89)-Bar/3
		Fourier		90	Fourier Axisym Shell (90)-Bar/3
		Isoparametric	15		Isoparametric Axisym Shell (15)-Bar/2
Planar Beam	Homogenous or Laminate	Standard Formulation	5	45	Planar Beam (5)-Bar/2, (45)-Bar/3
		Parabolic Shear Strain		45	Planar Beam w/Parabolic Strain (45)-Bar/3
	Curved Isoparametric		16		2D Curved Isoparametric Beam (16)-Bar/2
Rigid Line (LBC)					Rigid Elements (Lbc)

1D Elements (Thermal)

	Laminate	Parabolic Shear Strain		45	Planar Beam w/Parabolic Strain (45)-Bar/3
Rigid Line (LBC)					Rigid Elements (Lbc)
Link	Conduction		36	65	Conduction Link (36)-Bar/2, (65)-Bar/3
	Convection/Radiation		36	65	Convection/Radiation Link (36)-Bar/2, (65)-

1D Elements (Coupled)

1D (coupled)	Elastic Beam	General Section		98		Euler Beam w/General Section (98)-Bar/2
		Arbitrary Section		98		Euler Beam w/Arbitrary Section (98)-Bar/2
	Truss			9	64	Truss (9)-Bar/2, (64)-Bar/3
	Spring			SPRING		Spring
	Damper			SPRING		Damper
	Gap	Fixed Direction		12		Fixed Directional Gap (12)-Bar/2
		True Distance		12		True Distance Gap (12)-Bar/2
	Axisym Shell	Homogenous or Laminate	Standard Formulation	1	89	Axisymmetric Shell (1)-Bar/2, (89)-Bar/3
			Isoparametric	15		Isoparametric Axisym Shell (15)-Bar/2
		Linear Temp Distr		88	87	Shell w/Linear Temp (88)-Bar/2, (87)-Bar/3 - Stucturally Rigid
		Quadratic Temp Distr		88	87	Shell w/Quadratic Temp (88)-Bar/2, (87)-Bar/3 - Structurally Rigid
	Planar Beam	Homogenous or Laminate			45	Planar Beam (5)-Bar/2, (45)-Bar/3
	Link	Conduction		36	65	Conduction Link (36)-Bar/2, (65)-Bar/3 - Structrually Rigid
		Convection/Radiation		36	65	Convection/Radiation Link (36)-Bar/2, (65)-Bar/3 - Structurally

2D Elements (Structural)

Dimension	Type	Option 1	Option 2	Quad/4	Tri3	Quad/8	Tri/6	Input Properties Form label:
2D	Thin Shell	Homogenous or Laminate		139	138	72	49	Thin Shell (139)-Quad/4, (138)-Tri/3, (72)-Quad/8, (49)-Tri/6
	Thick Shell	Homogenous or Laminate	Standard Formulation	75	75	22	22	Thick Shell (75)-Quad/4-Tri/3, (22)-Quad/8-Tri/6
			Parabolic Shear Strain	75	75	22	22	Shell w/Parabolic Strain (75)-Quad/4-Tri/3, (22)-Quad/8-Tri6
			Reduced Integration	140	140			Shell w/Reduced Integration (140)-Quad/4-Tri/3
	Plate [1]			Remove				
	2D Solid	Plane Strain	Standard Formulation	11	6	27	125	Plane Strain (11)-Quad/4, (6)-Tri/3, (27)-Quad/8, (125)-Tri/6
			Assumed Strain [6]	Remove				
			Constant Volume	Remove				
			Assumed Strain/Constant Volume	Remove				
			Herrmann[2]	80	155	32	128	Herrmann Plane Strain (80)-Quad/4/5, (155)-Tri/3/4, (32)-Quad/8, (128)-Tri/6
			Herrmann/Reduced Integration	118	118	58	58	Herrmann/Reduced Integration Plane Strain (118)-Quad/4-Tri/3, (58)-Quad/8-Tri/6
			Reduced Integration	115	115	54	54	Reduced Plane Strain (115)-Quad/4-Tri/3, (54)-Quad/8-Tri/6
			Generalized	19	19	29	29	Generalized Plane Strain (19)-Quad/4-Tri/3, (29)-Quad/8-Tri/6
			Generalized/Constant Volume	Remove				
			Generalized/Reduced Integration			56	56	General/Reduced Plane Strain (56)-Quad/8-Tri/3
			Generalized/Herrmann	81	81	34	34	General/Herrmann Plane Strain (81)-Quad/4/5-Tri/3/4, (34)-Quad/8-Tri/6
			General/Herrmann/Red. Int.			60	60	General/Herrmann/Reduced Plane Strain (60)-Quad/8-Tri/6
			Laminated Composite	151	151	153	153	Laminated Composite Plane Strain (151)-Quad/4-Tri/3, (153)-Quad/8-Tri/6
			Semi-Infinite	91		93		Semi-Infinite Plane Strain (91)-Quad/4, (93)-Quad/8/9
		Plane Stress	Standard Formulation	3	3	26	124	Plane Stress (3)-Quad/4-Tri/6, (26)-Quad/8, (124)-Tri/6
			Assumed Strain	Remove				
			Reduced Integration	114	114	53	53	Reduced Plane Stress (114)-Quad/4-Tri/3, (53)-Quad/8-Tri/6
		Axisymmetric	Standard Formulation	10	2	28	126	Axisymmetric Solid (10)-Quad/4, (2)-Tri/3, (28)-Quad/8, (126)-Tri/6
			Constant Volume	Remove				
			Constant Volume/Twist	Remove				
			Herrmann	82	156	33	129	Herrmann Axisym Solid (82)-Quad/4/5, (156)-Tri/3/4, (33)-Quad/8, (129)-Tri/6
			Herrmann/Reduced Integration	119	156	59	59	Herrmann/Reduced Axisym Solid (119)-Quad/4/5, (156)-Tri/3/4, (59)-Quad/8-Tri/6
			Herrmann/Twist	83	83	66	66	Axisym Solid w/Twist (83)-Quad/4-Tri/3, (66)-Quad/8-Tri/6
			Reduced Integration	116	116	55	55	Reduced Axisym Solid (116)-Quad/4-Tri/3, (55)-Quad/8-Tri/6
			Twist	20	20	67	67	Axisym Solid w/Twist (20)-Quad/4-Tri/3, (67)-Quad/8-Tri/6
			Fourier			62	62	Fourier Axisym Solid (62)-Quad/8-Tri/6
			Herrmann/Fourier			63	63	Herrmann/Fourier Axisym Solid (63)-Quad/8-Tri/6
			Reduced Integration/Fourier			73	73	Reduced/Fourier Axisym Solid (73)-Quad/8-Tri/6
			Herrmann/Red. Int./Fourier			74	74	Herrmann/Reduced/Fourier Axisym Solid (74)-Quad/8-Tri/6
			Bending	95	95	96	96	Axisym Solid w/Bending (95)-Quad/4-Tri/3, (96)-Quad/8-Tri/6
			Laminated Composite	152	152	154	154	Laminated Composite Axisym Solid (152)-Quad/4-Tri/3, (154)-Quad/8-Tri/6
			Semi-Infinite[2]	92		94		Semi-Infinite Axisym Solid (92)-Quad/4, (94)-Quad/8/9
	Membrane			18	18	30		Membrane (18)-Quad/4-Tri/3, (30)-Quad/8
	Shear Panel			68				Shear Panel (68)-Quad/4
	Rebar	Plane Strain	Standard Formulation	143		46		Plane Strain w/Rebar (143)-Quad/4, (46)-Quad/8
			Generalized			47		Generalized Plane Strain w/Rebar (47)-Quad/8
		Axisymmetric	Standard Formulation	144		48		Axisymmetric Solid w/Rebar - (144)-Quad/4, (48)-Quad/8
			Twist	145		142		Axisym Solid w/Rebar and Twist - (145)-Quad/4, (48)-Quad/8
		Membrane		147		148		Membrane w/Rebar (147)-Quad/4, (148)-Quad/8
	Rigid Surface (LBC)							Rigid Elements (Lbc)

2D Elements (Coupled)

2D (coupled)	Thin Shell	Homogenous or Laminate		139	138	72	49	Thin Shell (139)-Quad/4, (138)-Tri/3, (72)-Quad/8, (49)-Tri/6
	Thick Shell	Homogenous or Laminate	Standard Formulation	75	75	22	22	Thick Shell (75)-Quad/4-Tri/3, (22)-Quad/8-Tri/6
			Parabolic Shear Strain	75	75	22	22	Shell w/Parabolic Strain (75)-Quad/4-Tri/3, (22)-Quad/8-Tri6
			Reduced Integration	140	140			Shell w/Reduced Integration (140)-Quad/4-Tri/3
	2D Solid	Plane Strain	Standard Formulation	11	6	27	125	Plane Strain (11)-Quad/4, (6)-Tri/3, (27)-Quad/8, (125)-Tri/6
			Herrmann[2]	80	155	32	128	Herrmann Plane Strain (80)-Quad/4/5, (155)-Tri/3/4, (32)-Quad/8, (128)-Tri/6
			Herrmann/Reduced Integration	118	118	58	58	Herrmann/Reduced Integration Plane Strain (118)-Quad/4-Tri/3, (58)-Quad/8-Tri/6
			Reduced Integration	115	115	54	54	Reduced Plane Strain (115)-Quad/4-Tri/3, (54)-Quad/8-Tri/6
			Generalized	19	19	29	29	Generalized Plane Strain (19)-Quad/4-Tri/3, (29)-Quad/8-Tri/6
			Generalized/Reduced Integration			56	56	General/Reduced Plane Strain (56)-Quad/8-Tri/3
			Generalized/Herrmann	81	81	34	34	General/Herrmann Plane Strain (81)-Quad/4/5-Tri/3/4, (34)-Quad/8-Tri/6
			General/Herrmann/Red. Int.			60	60	General/Herrmann/Reduced Plane Strain (60)-Quad/8-Tri/6
			Semi-Infinite	91		93		Semi-Infinite Plane Strain (91)-Quad/4, (93)-Quad/8/9
		Plane Stress	Standard Formulation	3	3	26	124	Plane Stress (3)-Quad/4-Tri/6, (26)-Quad/8, (124)-Tri/6
			Reduced Integration	114	114	53	53	Reduced Plane Stress (114)-Quad/4-Tri/3, (53)-Quad/8-Tri/6
		Axisymmetric	Standard Formulation	10	2	28	126	Axisymmetric Solid (10)-Quad/4, (2)-Tri/3, (28)-Quad/8, (126)-Tri/6
			Constant Volume/Twist	20	20			Constant Axisymmetric Solid (20)-Quad/4
			Herrmann	82	156	33	129	Herrmann Axisym Solid (82)-Quad/4/5, (156)-Tri/3/4, (33)-Quad/8, (129)-Tri/6
			Herrmann/Reduced Integration	119	156	59	59	Herrmann/Reduced Axisym Solid (119)-Quad/4/5, (156)-Tri/3/4, (59)-Quad/8-Tri/6
			Herrmann/Twist	83	83	66	66	Axisym Solid w/Twist (83)-Quad/4-Tri/3, (66)-Quad/8-Tri/6
			Reduced Integration	116	116	55	55	Reduced Axisym Solid (116)-Quad/4-Tri/3, (55)-Quad/8-Tri/6
			Twist	20	20	67	67	Axisym Solid w/Twist (20)-Quad/4-Tri/3, (67)-Quad/8-Tri/6
			Semi-Infinite[2]	92		94		Semi-Infinite Axisym Solid (92)-Quad/4, (94)-Quad/8/9
	Membrane			18	18	30		Membrane (18)-Quad/4-Tri/3, (30)-Quad/8
	Thermal Shell	Homogenous or Laminate	Linear Temp Distr	85	50	86	86	Shell w/Linear Temp (85)-Quad/4, (50)-Tri/3, (86)-Quad/8-Tri/6 - Structurally
			Quadratic Temp Distr	85	50	86	86	Shell w/Linear Temp (85)-Quad/4, (50)-Tri/3, (86)-Quad/8-Tri/6 - Structurally
	Thermal 2D Solid	Planar	Standard Formulation	39	37	41	131	Planar Solid (39)-Quad/4, (37)-Tri/6, (41)-Quad/8, (131)-Tri/6 - Structurally
			Reduced Integration	121	121	69	69	Reduced Planar Solid (121)-Quad/4-Tri/6, (69)-Quad/8-Tri/6- Structurally
			Semi-Infinite	101		103		Semi-Infinite Planar Solid (101)-Quad/4, (103)-Quad/8/9 - Structurally Rigid
		Axisymmetric	Standard Formulation	40	38	42	132	Axisym Solid (40)-Quad/4, (38)-Tri/3, (42)-Quad/8, (132)-Tri/6
			Reduced Integration	122	122	70	70	Reduced Axisym Solid (122)-Quad/4-Tri/3, (70)-Quad/8-Tri/6
			Semi-Infinite	102		104		Semi-Infinite Axisym Solid (102)-Quad/4, (104)-Quad/8/9

2D Elements (Electromagnetic)

2D (electromagnetic)	2D Solid	Planar	111				Planar Electromagnetic (111)-Quad/4
		Axisymmetric	112				Axisymmetric Electromagnetic (112)-Quad/4

3D Elements (Structural)

Dimension	Type	Option 1	Option 2	Hex/8	Wed/6	Tet/4	Hex/20	Wed/15	Tet/10	Input Properties Form label:
3D	Solid	Standard Geometry	Standard Formulation	7	7	134	21	21	127	Solid (7)-Hex/8-Wed/6, (134)-Tet/4, (21)-Hex/20-Wed/15
			Assumed Strain	Remove						
			Constant Volume	Remove						
			Constant Volume/Assumed Strain	Remove						
			Herrmann[2]	84	84	157	35	35	130	Herrmann Solid (84)-Hex/8/9-Wed/6/7, (157)-Tet/4/5, (3
			Herrmann/Reduced Integration	120	120	157	61	61	130	Herrmann/Reduced (120)-Hex/8/9-Wed/6/7, (157)-Tet/4 Tet/10
			Reduced Integration	117	117	134	57	57	127	Reduced Solid (117)-Hex/8-Wed/6, (134)-Tet/4, (57)-He
		Auto Tying Shell	Standard Formulation		7			21		Solid w/Auto Tie (7)-Wed/6, (21)-Wed/15
			Assumed Strain	Remove						
			Constant Volume	Remove						
			Constant Volume/Assumed Strain	Remove						
			Reduced Integration					57		Reduced Solid w/Auto Tie (57)-Wed/15
		Laminated Composite		149	149		150	150		3D Laminated Composite (149)-Hex/8-Wed/6, (150)-He
		Rebar		146			23			3D Rebar (146)-Hex/8, (23)-Hex/20
		Semi-Infinite		107			108			3D Semi-Infiite (107)-Hex/8, (108)-Hex/20/27

3D Elements (Thermal)

3D (thermal)	Solid	Standard Formulation	43	43	135	44	44	133	Solid (43)-Hex/8-Wed/6, (135)-Tet/4, (44)-Hex/20-Wed/1
		Reduced Integration	123	123	135	71	71	133	Reduced Solid (123)-Hex/8-Wed/6, (135)-Tet/4, (71)-He
		Semi-Infinite	105			106			3D Semi-Infiite (105)-Hex/8, (106)-Hex/20/27

3D Elements (Coupled)

3D (coupled)	Solid	Standard	Standard Formulation	7	7	134	21	21	127	Solid (7)-Hex/8-Wed/6, (134)-Tet/4, (21)-Hex/20-Wed/15
			Herrmann[2]	84	84	157	35	35	130	Herrmann Solid (84)-Hex/8/9-Wed/6/7, (157)-Tet/4/5, (35
			Herrmann/Reduced Integration	120	120	157	61	61	130	Herrmann/Reduced (120)-Hex/8/9-Wed/6/7, (157)-Tet/4/ Tet/10
			Reduced Integration	117	117	134	57	57	127	Reduced Solid (117)-Hex/8-Wed/6, (134)-Tet/4, (57)-Hex
		Auto Tying	Standard Formulation		7			21		Solid w/Auto Tie (7)-Wed/6, (21)-Wed/15
			Reduced Integration					57		Reduced Solid w/Auto Tie (57)-Wed/15
		Semi-Infinite		107			108			3D Semi-Infiite (107)-Hex/8, (108)-Hex/20/27
	Thermal Solid	Standard Formulation		43	43	135	44	44	133	Solid (43)-Hex/8-Wed/6, (135)-Tet/4, (44)-Hex/20-Wed/1
		Reduced Integration		123	123	135	71	71	133	Reduced Solid (123)-Hex/8-Wed/6, (135)-Tet/4, (71)-He
		Semi-Infinite		105			106			3D Semi-Infiite (105)-Hex/8, (106)-Hex/20/27 - Structura

3D Elements (Magnetostatic)

3D (magnetostatic)	Solid	Standard Formulation	109						Magnetostatic Solid (109)-Hex/8
		Semi-Infinite	110						Magnetostatic Solid (110)-Hex/8

3D Elements (Electromagnetic)

3D (electromagnetic)	Solid	113						Electromagnetic Solid (113)-Hex/8

Figure 5. Supported Material Types

Isotropic Material Properties

Constitutive Model					Comments
Elastic					Creates **ISOTROPIC** option
Constitutive Model	**Type**	**Hardening Rule**	**Yield Criteria**	**Strain Rate Method**	**Comments**
Plastic	Elastic-Plastic	Isotropic	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	Creates **ISOTROPIC, WORK HARD** and **STRAIN RATE** options
		Kinematic	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	Creates **ISOTROPIC, WORK HARD** and **STRAIN RATE** options
		Combined	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	Creates **ISOTROPIC, WORK HARD** and **STRAIN RATE** options
		Power Law			Creates only **ISOTROPIC** option w/ power
		Rate Power Law			Creates only **ISOTROPIC** option w/ rate
		Johnson-Cook			Creates only **ISOTROPIC** option w/
		Kumar			Creates only **ISOTROPIC** option w/ Kumar
	Perfectly Plastic	None	Same as E-P Isotropic or Kinematic or Combined		Creates only **ISOTROPIC** option w/ equivalent yield stress and **STRAIN RATE**
	Rigid-Plastic	Piecewise Linear	None	Piecewise Linear Cowper-Symonds	Creates only **ISOTROPIC** option w/ no E or nu from the **Elastic** constitutive model or any yield criteria
		Power Law			
		Rate Power Law			
		Johnson-Cook			
		Kumar			
	User Subroutine (WKSLP)	Isotropic	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	Same as Elastic-Plastic
		Kinematic	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	
		Combined	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	
	User Subroutine (URPFLO)				Same as Rigid-Plastic

Isotropic Material Properties (Continued)

Constitutive Model	Failure Criteria	Comments
Failure	Hill Hoffman Tsai-Wu Maximum Strain Maximum Stress User Subroutine (UFAIL)	No change in functionality except UFAIL added

Constitutive Model	Model	Domain Type	Number of Terms	Comments
Hyperelastic	Neo-Hookean	Time Frequency	1	Creates **MOONEY** option w/ only C10 and **PHI-COEFFICIENTS** for freq domain
	Mooney-Rivlin	Time Frequency	1	Creates **MOONEY** option w/ only C10/C01 and **PHI-COEFFICIENTS** for freq domain
	Jamus-Green-Simpson	Time Frequency	1	Creates **MOONEY** options w/ all Cs and **PHI-COEFFICIENTS** for freq domain
	Ogden	Time	1 2 3 4 5 6	Creates **OGDEN** option
	Foam	Time	1 2 3 4 5 6	Creates **FOAM** option
	Arruda-Boyce	Time	1	Creates **ARRUDBOYCE** option
	Gent	Time	1	Creates **GENT** option

Constitutive Model	Comments
Viscoelastic	Creates **VISCEL EXP**, **VISCEPROP, VISCELMOON,** or **VISCELOGDEN** depending on properties supplied
Creep	Creates **CREEP** option
Damping	Creates **DAMPING** option

2D/3D Orthotropic Material Properties

Constitutive Model	2D Condition	Comments
Elastic	Plane Stress/Thin Shell Plane Strain/Axisymmetric Thick Shell Axisymmetric with Twist Axisymmetric Shell	Creates **ORTHOTROPIC** option
	None (3D Orthotropic Case)	

Constitutive Model	Type	Hardening Rule	Yield Criteria	Strain Rate Method	Comments
Plastic	Elastic-Plastic	Isotropic	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	Creates **ORTHOTROPIC, WORK HARD** and **STRAIN RATE** options
		Kinematic	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	Creates **ORTHOTROPIC, WORK HARD** and **STRAIN RATE** options
		Combined	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	Creates **ORTHOTROPIC, WORK HARD** and **STRAIN RATE** options
	Perfectly Plastic	None	Same as E-P Isotropic or Kinematic or Combined		Creates only **ORTHOTROPIC** option w/ equivalent yield stress and **STRAIN RATE**
	User Subroutine (WKSLP)	Isotropic	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	Same as Elastic-Plastic
		Kinematic	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	
		Combined	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	

Constitutive Model	Failure Criteria	Comments
Failure	Hill Hoffman Tsai-Wu Maximum Strain Maximum Stress User Subroutine (UFAIL)	No change in functionality except UFAIL added, but needs to be added to 3D Orthotropic

Constitutive Model	Comments
Viscoelastic	Creates **VISCEL EXP** or **VISCELORTH** depending on properties supplied
Creep	Creates **CREEP** option
Damping	Creates **DAMPING** option

2D/3D Anisotropic Material Properties

Constitutive Model	2D Condition	Comments
Elastic	Plane Stress/Thin Shell Plane Strain/Axisymmetric Thick Shell Axisymmetric with Twist Axisymmetric Shell	Creates **ANISOTROPIC** option
	None (3D Orthotropic Case)	

Constitutive Model	Type	Hardening Rule	Yield Criteria	Strain Rate Method	Comments
Plastic	Elastic-Plastic	Isotropic	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	Creates **ANISOTROPIC, WORK HARD** and **STRAIN RATE** options
		Kinematic	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	Creates **ANISOTROPIC, WORK HARD** and **STRAIN RATE** options
		Combined	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	Creates **ANISOTROPIC, WORK HARD** and **STRAIN RATE** options
	Perfectly Plastic	None	Same as E-P Isotropic or Kinematic or Combined		Creates
	User Subroutine (WKSLP)	Isotropic	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	Same as Elastic-Plastic
		Kinematic	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	
		Combined	von Mises Linear Mohr-Coulomb Parabolic Mohr Coulomb Buyukozturk Concrete Oak Ridge National Lab 2-1/4 Cr-Mo ORNL Reversed Plasticity ORNL Full Alpha Reset ORNL Generalized Plasticity	Piecewise Linear Cowper-Symonds	

REFERENCES

(1) *MSC.Marc Preference Software Design Document*, The MSC Software Corporation, Los Angeles, CA , March 2000.