

TrueGrid® Output Manual For DYNA3D

A Guide and a Reference

by

Robert Rainsberger

Version 2.3.0

XYZ Scientific Applications, Inc.

October 12, 2006

Copyright © 2006 by XYZ Scientific Applications, Inc. All rights reserved.

TrueGrid,® the **TrueGrid**® Output Manual for DYNA3D, and related products of XYZ Scientific Applications, Inc. are copyrighted and distributed under license agreements. Under copyright laws, they may not be copied in whole or in part without prior written approval from XYZ Scientific Applications, Inc. The license agreements further restrict use and redistribution.

XYZ Scientific Applications, Inc. makes no warranty regarding its products or their use, and reserves the right to change its products without notice. This manual is for informational purposes only, and does not represent a commitment by XYZ Scientific Applications, Inc. XYZ Scientific Applications, Inc. accepts no responsibility or liability for any errors or inaccuracies in this document or any of its products.

TrueGrid® is a registered trademark of XYZ Scientific Applications, Inc.

Some other product names appearing in this book may also be trademarks or registered trademarks of their trademark holders.

Table of Contents

Table of Contents	3
I. DYNA3D Output Guide	5
Introduction	5
Font Conventions	5
Supported Features	5
Sliding (or Contact) Surfaces	7
Initial and Boundary Conditions	8
Loads	9
Load Curves	10
Stone Walls	11
Bricks	11
Shells	11
Thick Shells	12
Beams	12
Joints and Rigid Bodies	13
Springs and Dampers and Point Masses	13
Temperatures	13
Shared Constraints	13
Tied with Failure	14
Post Processing	14
II. DYNA3D Output Example	15
III. DYNA3D Output Reference	19
Command Syntax Conventions	19
bsd global beam cross section definition	19
sid sliding interface definition	21
si select nodes or faces for sliding interface in merge phase	23
si assign sliding interface to region in part phase	23
sii assign sliding interfaces to progression in part phase	23
spd define the properties of a set of springs or dampers	25
dynaeos DYNA3D equation of state	26
dynamats DYNA3D materials	30
dynaopts DYNA3D analysis options	47
IV. Frequently Asked Questions	59
V. INDEX	61

I. DYNA3D Output Guide

Introduction

DYNA3D is a nonlinear, explicit, three-dimensional finite element code for solid and structural mechanics developed at Lawrence Livermore National Laboratories. The focus in this manual will be on those features in **TrueGrid®** that are specific to creating a DYNA3D input file. The **TrueGrid®** User Manual covers the creation of a mesh and will not be covered in this manual. This manual is incomplete in another way because it cannot be used as a substitute for the DYNA3D manual. For a full understanding of the use of these features, the user must have a working knowledge of DYNA3D and be familiar with a DYNA3D User Manual and, in particular, the one written by Jerry I. Lin, dated January, 2005, UCRL-MA-107254.

Font Conventions

Different fonts are used through out this manual to indicate their meaning. A literal is highlighted in bold. A symbol to be substituted with a literal or a number is *italicized*. A computer example uses the *Courier* font.

Supported Features

There are many features in **TrueGrid®** to create a model for DYNA3D. The table below shows the commands that are used for each feature. Sometimes there are several commands listed. For example, shells can be generated using both the **block** and **cylinder** commands. The **n** and **th** are used to set the properties of these shells. In another example, the **si** and **sii** commands are used to identify the faces of the mesh that form the sliding (or contact) surfaces. The associated **sid** command is used to assign properties to the sliding surface.

<u>DYNA3D feature</u>	<u>TrueGrid® commands</u>
parameters in the control cards	dynaopts
materials	dynamats, mate, mt, mti
equation of state	dynaeos
truss and beam element cross section properties	dynamats, bsd
shell element cross section properties	dynamats
thick shell element cross section properties	dynamats
beam user defined integration rules	bind
shell user defined integration rules	sind
delamination elements	(under development)
cohesive elements	(under development)
nodal boundary conditions	b, bi

symmetry
solid elements
beam and truss elements
shell and membrane elements
thick shell elements
interface save segment definition
nodal arbitrary single point constraints
sliding boundary planes
symmetry planes with failure
node time history blocks
element time history blocks
gravity stress initialization
brode functions
cross section definitions for force output
load curves
nodal forces and follower forces
pressure loads
prescribed velocities/accelerations/displacements
rigid or stone walls
coupled or shared nodal constraints
spot welds
rigid node set
initial velocity conditions
material initial rotation
sliding (or contact) surface
tie-breaking shell slide line
tied node sets with failure
rigid body merges
extra nodes with rigid bodies
rigid body joints
prescribed base accelerations
prescribed angular velocities
momentum deposition in solid elements
detonation points
shell-solid interfaces
discrete springs, dampers, and masses
rigid body inertial properties
nonreflecting boundary segments
temperature input option I
temperature input option II
one dimensional slide line
material initialization for rotational motion

plane
block, cylinder, or
block, cylinder, ibm, jbm, kbm, bm
block, cylinder, n, th, thi, ssf, ssfi, or
block, cylinder, or
iss
lsys, lb, sfb
plane, sfb
plane, syf
npb
epb
dynaopts (gvst)
dynaopts
csf
lcd, flcd
fc, mom, ndl, ffc, fmom
pr, pramp, dom, arri, dist
fv, acc, fd, frb, dynamats (bpm)
sw, swi
mpc
(under development)
(under development)
rotation, velocity, ve
(under development)
sid, si, sii
(under development)
fn, fni
rigbm
jt
jd, jt
dynaopts (grav)
dynaopts (xvel, yvel, zvel)
mdep
detp
(under development)
spd, spdp, spring, pm, npm
(under development)
nr, nri
tepro
temp, te, tei
sid, si, sii
(under development)

body force by material	(under development)
CVS (MADTMO/ATB) Coupling	(under development)
Air Bag Gas Flow Definitions	(under development)
slide surface activation/deactivation times	(under development)
fiber orientation	(under development)
set the problem title	title
comments for specific sections of the output	comment
merge parts into a single connected model	stp, tp, t
view properties in the merge phase	co
select the DYNA3D output format	dyna3d
write a DYNA3D input file	write

You may want to view some of the properties graphically using the **condition (co)** command in the merge phase. The **tmm** command can be used to calculate the mass of each part. Be sure to merge the nodes using one of the merging commands such as **stp** and, finally, use the **dyna3d** command to select DYNA3D as the output option and the **write** command to actually create the input deck for DYNA3D.

The file produced by **TrueGrid®** is an ASCII file that can be examined or modified using any text editor. Some experienced users always inspect the file and modify it instead to rerunning **TrueGrid®** when make easy changes to the model such as changing a material model parameter or the time step. For this reason, the output file has helpful comments. However, this file can be very large and it might be easier to modify the **TrueGrid®** session file and rerun **TrueGrid®**. One of the comments that is automatically written has a time stamp for archiving purposes. The **title** is also helpful for archiving. You can insert your own comments with the use of the **comment** command.

The **readmesh** command has a **dyna3d** option so that you can import a DYNA3D model into **TrueGrid®**. This is intended to be used to translate a DYNA3D input file into another format or to make small modifications to a model when a session file does not exist. This feature does not replace the session file because the block structure of the mesh cannot be reconstructed. This also means there are no block boundary interfaces (**bb** command) to utilize. If there is a block structure underlying the mesh, you can form a block boundary interface using the **mbb** command, but this can be tedious. As a cautionary note, because DYNA3D has header data that prescribe the bulk data that follows it in the input file, when the format is changed due to additions, until the **readmesh** has been updated, the additional data in the DYNA3D file will cause errors in the **readmesh** command. You may have to experiment or check the documentation on **readmesh** to determine which features are supported in **readmesh**.

Sliding (or Contact) Surfaces

To form a contact surface, use the **sid** command to define the surface type. Some types have only one side. Some are formed from faces of bricks or shells. Others are formed partially from nodes. The

sid command also has optional parameters such as friction. Be sure to choose a DYNA3D type, since other types will not be recognized when writing the output file.

While in the part phase use the **si** or **sii** commands to select faces of that part for inclusion in the surface definition. If the face is from a shell element, be sure to use the **orpt** orientation command prior to issuing the **si** or **sii** command so that the orientation of the face is towards the opposing face in the sliding surface. If you are using part replication (**lrep**, **grep**, or **pslv**), then you may want to use the **lsii** or the **gsii** to increment the sliding interface command for each replication. You must use the **sid** command for each sliding surface that is referenced when the **lsii** or **gsii** commands are used with replication.

You can use sets in the merge phase to add faces or nodes to a sliding surface. These sets can be formed with the combined use of the **fset** (for faces) and the **nset** (for nodes) commands in the part and merge phase. Only use node sets when defining a sliding surface where nodes are on the slave side and otherwise only use face sets. The node density between the master and slave sides of the interface should be roughly equal. When forming the mesh in the part phase, it may be necessary to build into the mesh a small gap between the master and slave sides of the contact surfaces, depending on the mesh density and the curvature to avoid initial penetration of the slave side into the master side.

When you merge the nodes (in the merge phase), the nodes from the slave side will not be allowed to merge with the nodes on the master side. Use the **mns** command in the merge phase to override this condition. When you first merge the nodes, a table will be printed to the text window and the **tsave** file listing the number of faces and nodes associated with each sliding surface. Check this table carefully. You can also see the faces and nodes of either side of the sliding surfaces using the **co** command. When using this in combination with the hide graphics option, you can see the orientation of the faces. Use **labels** command to show how the nodes have merged graphically.

Initial and Boundary Conditions

There are several ways to constrain nodes. The **b** and **bi** commands in the part phase or the **b** command in the merge phase will constrain nodes in the global coordinate system. Use the **plane** command to specify symmetry plane constraints including symmetry planes with failure. Nodes in the model will be assigned to these symmetry planes based on the tolerance you specify in the **plane** command. The **lb** (and the associated **lsys**) command can be used to set the constraints in any coordinate system. The **sfb** command can also be used to do this. Be sure that something in the model has been constrained or the entire model might fly off.

To set non-reflective (or transmitting) boundary conditions, use the **nr** and **nri** commands in the part phase or the **nr** command in the merge phase. Special care is needed when developing a model using this type of boundary condition. See the DYNA3D User Manual for details.

If you use the **velocity** or **rotation** command in the control phase, then all subsequent parts will be assigned this initial velocity. This can be over ridden using the **velocity** or **rotation** command within a part. Both of these conditions can be over ridden for specific regions of the mesh using the **ve** or **vei** commands in the part phase or the **ve** command in the merge phase. Velocities are not accumulative. Care is needed when assigning initial velocities so that when two nodes are merged, the velocities of those two nodes match. Only one of the velocities will be used and if they do not match, you may get an unexpect result. Usually, if the velocities of two merged nodes do not match, this indicates an error in the model. There are other concerns regarding incompatible initial velocities and prescriptions and the DYNA3D User Manual discusses this issue. **TrueGrid®** does not protect you from or identify these incompatibilities.

Loads

There are numerous ways to assign loads. The list of commands that can be used to assign loads in the part phase includes:

fc	Cartesian concentrated nodal loads
fci	Cartesian concentrated nodal loads
fcc	cylindrical concentrated nodal loads
fccci	cylindrical concentrated nodal loads
fcs	spherical concentrated nodal loads
fcsi	spherical concentrated nodal loads
mom	nodal moment about one of the nodal axis in the global coordinate system
momi	nodal moment about one of the nodal axis in the global coordinate system
ndl	pressure converted to distributed nodal loads
ndli	pressure converted to distributed nodal loads
pr	pressure loads on element faces
pri	pressure loads on element faces
pramp	pressure loads on element faces
fv	Cartesian prescribed nodal velocities
fvi	Cartesian prescribed nodal velocities
fvc	cylindrical prescribed nodal velocities
fvcii	cylindrical prescribed nodal velocities
fvs	spherical prescribed nodal velocities
fysi	spherical prescribed nodal velocities
fvv	Cartesian variable prescribed nodal velocities
fvvi	Cartesian variable prescribed nodal velocities
fvvc	cylindrical variable prescribed nodal velocities
fvvcii	cylindrical variable prescribed nodal velocities
fvvs	spherical variable prescribed nodal velocities
fvysi	spherical variable prescribed nodal velocities
acc	Cartesian prescribed nodal acceleration

acci	Cartesian prescribed nodal acceleration
accc	cylindrical prescribed nodal acceleration
accci	cylindrical prescribed nodal acceleration
accs	spherical prescribed nodal acceleration
accsi	spherical prescribed nodal acceleration
vacc	Cartesian variable prescribed nodal acceleration
vacci	Cartesian variable prescribed nodal acceleration
vacc	cylindrical variable prescribed nodal acceleration
vacci	cylindrical variable prescribed nodal acceleration
vaccs	spherical variable prescribed nodal acceleration
vacci	spherical variable prescribed nodal acceleration
fd	Cartesian displacement
fdi	Cartesian displacement
fdc	cylindrical displacement
fdci	cylindrical displacement
fds	spherical displacement
fdsi	spherical displacement
frb	prescribed rotation
frbi	prescribed rotation

The list of commands that can be used to assign loads in the merge phase includes:

fc	Cartesian concentrated nodal loads
mom	nodal moment about one of the nodal axis in the global coordinate system
ndl	pressure converted to distributed nodal loads
pr	pressure loads on element faces
pramp	pressure loads on element faces
fv	Cartesian prescribed nodal velocities
fvv	Cartesian variable prescribed nodal velocities
vacc	Cartesian variable prescribed nodal acceleration
fd	Cartesian displacement
frb	prescribed rotation
ffc	concentrated nodal load with a follower force
fmom	nodal moment with a follower force

The **pramp** command is used with either **pr** or **pri**. It applies a pressure based on a function for all nodes that have a zero pressure. In most cases, the magnitude of the load is specified using a load curve. This varies the amplitude of the load with respect to time.

Load Curves

Load curves are 2D polygonal curves that can be created using the **lcd** and **flcd** commands. Load curves are typically used to define the relative amplitude of a load with respect to time. They can be used to relate any two variables. Almost all prescribed loads require a load curve in time so that the amplitude of the load can vary. It is best to define a load curve before it is referenced in a load or material model to avoid a warning message. When the output file is written, if a load curve is referenced but not defined, you will also receive a warning message. Then a simple load curve will be used in the output file so that a valid DYNA3D file is written. It is advised that you correct this by defining the appropriate load curve for the problem. Do not rely on the load curve that is automatically generated.

In some dialogue boxes you might be prompted for a load curve or a set id. This is because such commands can be used to define, for example, a dynamic load for DYNA3D or a static load for another output option that has the option to turn loads on or off depending on the set id. Simply ignore the set id portion of the prompt and supply the load curve number.

Stone Walls

A stone wall is defined with two commands. Use the **plane** command to set the properties of the stone wall. Nodes to react to the stone wall will not be selected automatically based on the tolerance. Use the **sw** and **swi** commands in the part phase to assign faces of the model to react to the stone wall. You can use the **sw** command in the merge phase as well to assign faces from a face set to react to the stone wall.

Bricks

Brick elements refer to hexahedral, prism (wedge), and tetrahedral elements and are considered the same type in DYNA3D. Only one element type can be associated with a material definition. If you want two different element types with the same material properties, you must define two materials. Most, but not all, materials support the different brick element types. There are no section properties for bricks. Be sure to use the **mate**, **mt**, or **mti** command to assign the proper material to each section of the mesh.

The element local coordinate system used in an orthotropic or anisotropic material is imposed by the order of the nodes that define the element. You can flip the nodal ordering to switch the orientation of this local coordinate system using the **or** command in the part phase.

Shells

Shell elements refer to both quadrilateral and triangular elements and sometimes referred to as structural elements. Cross sectional properties are included in the material model when the shell type is selected. There are no section properties for bricks. Be sure to use the **mate**, **mt**, or **mti** command to assign the proper material to each section of the mesh. The default shell thicknesses are included

as part of the cross sectional properties. These default thicknesses can be over ridden with the use of the **thic** command in the part phase. Both can be over ridden for a region of the part using the **th** and **thi** commands. If you have two surfaces that represent the inner and outer surfaces of a structure that is to be modeled using shell elements, than you can use the **ssf** and **ssf1** commands in the part phase to create shells with variable thickness.

The orientation of the positive normal direction to the shell is dictated by the nodal ordering of the nodes that define the shell. This positive direction is used, for example, to determine the direction of a positive pressure. This direction can be flipped using the **n** command in the part phase. The order of the nodes also dictate the local material coordinate system which can be important when using an orthotropic or anisotropic material. Use the **or** command to flip the coordinate system to the desired direction. When an angle is specified for the orientation of a composite material, it is with respect to this orientation.

You may need to specify the through thickness integration points when defining a composite material. This can be done by defining an integration rule with the **sind** command. Then identify this rule in the material definition.

Thick Shells

Thick shells are generated and look like hexahedral elements. Only a few materials support thick shells. Be sure to use the **mate**, **mt**, or **mti** command to assign the proper material to each section of the mesh.

The element local coordinate system used in an orthotropic or anisotropic material is imposed by the order of the nodes that define the element. You can flip the nodal ordering to switch the orientation of this local coordinate system using the **or** command in the part phase.

You may need to specify the through thickness integration points when defining a composite material. This can be done by defining an integration rule with the **sind** command. Then identify this rule in the material definition. All other cross section information is specified in the material definition.

Beams

Two nodes are required to form a beam element. In many cases, a third node is needed to define the local coordinate system used to form the cross sectional properties. These element are sometimes referred to as structural elements. Use the **ibm**, **ibmi**, **jbm**, **jbmi**, **kbm**, and **kbmi** commands to form beam elements with shell or brick structures while in the part phase. If the material of the shell or brick structure is set to zero using the **mt**, **mti**, or **mate** command, then the shells or bricks will be ignored, but the embedded beams will not be ignored. This is a convenient way to build an array of beams using block structured methods. You can also use the **bm** command in the merge phase to

build a string of beams that can be made to follow a 3D curve. The **beam** command (this command has been denigrated) can also be used to form beam elements, but the command is not interactive.

Both the element type and the default cross section properties are defined in the material definition. You can also use the **bsd** command to define cross sectional properties to over ride the material default cross sectional properties. When you create a beam, refer to the **bsd** number to assign these cross sectional properties to the beam. Use the **bind** command to define beam integration rule, if needed. Then refer to this integration rule when defining the material.

Joints and Rigid Bodies

A rigid body is formed using shells and bricks that are assigned the rigid body material. Each rigid body can be attached to other parts of the model using joints. A joint is defined in two steps. The **jd** command is used to define the properties of a joint. Then the **jt** command is used to identify which nodes are used to form the joint. Nodes within a joint are not merged.

Springs and Dampers and Point Masses

Springs and dampers are treated the same in **TrueGrid®**. They are only distinguishable by the material properties assigned them. Use the **spd** command to define the properties of the spring or damper. Then use the **spring** command to assign nodes to a numbered spring. Alternatively, the **spdp** command can be used in the part phase to create an array of springs between two parts, analogous to a contact surface.

Point masses can be generated in the part or merge phase. There are two types of point masses. The **pm** command will assign a mass to an existing node. The **npm** will create a new node and assign it a mass. The latter must then be connected either to a spring or beam.

Temperatures

There are two methods in DYNA3D to set the temperatures for material properties. For the first option, use the **tepro** command in the part or merge phase. You can use the **temp** command to set a default constant temperature. Then use the **te** and **tei** commands in the part phase or the **te** command in the merge phase to vary the temperature in different regions of the mesh.

Shared Constraints

Use the **mpc** command to couple a set of nodes. This requires that you create a node set first. The **nset** or **nseti** command can be used in the part phase and the **nset** command in the merge phase to create a node set. Also, click on the pick button in the environment window during the merge phase. Then you can use the mouse to modify or create a node set. The nodes sharing a set of constraints will not be merged together.

Tied with Failure

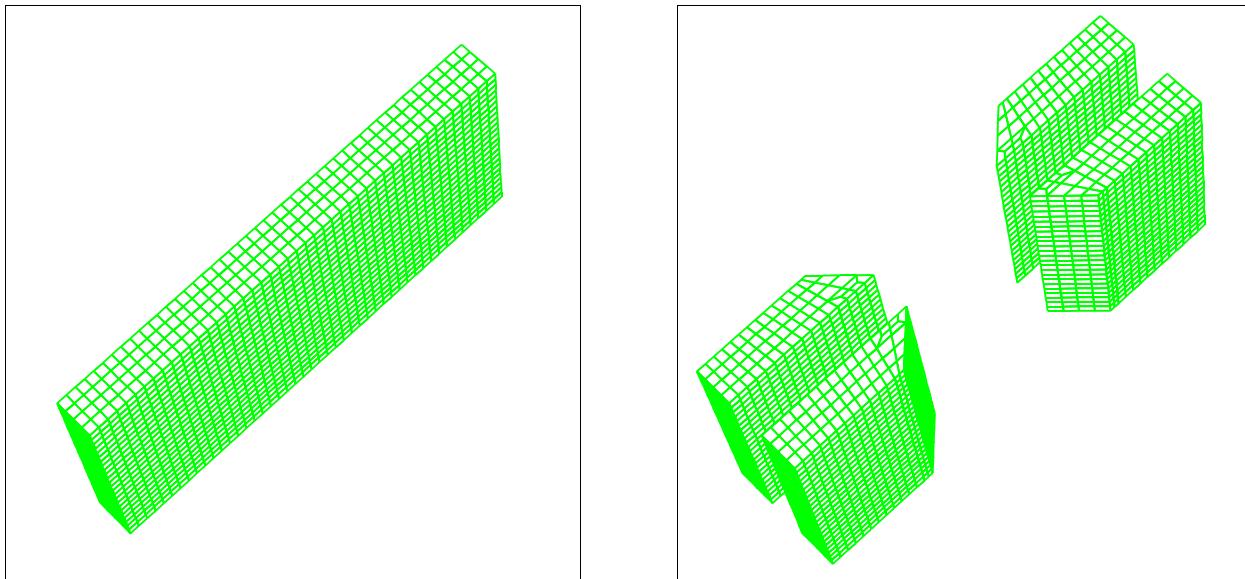
The **fn** and **fni** commands in the part phase will generate a shell mesh where there are 4 unique nodes for each shell element. These are automatically tied together with a failure criteria where shell elements meet. The nodes that are tied together are merged together in the merge phase.

Post Processing

There are a number of options of the **dynaopts** command to control the data saved in the database by DYNA3D for post processing. You can get more information from the reference section on the **iff**, **prt1**, **plt1**, **ssdm**, **drtflg** options of **dynaopts**. You may wish to analyze in greater detail the evolution of certain nodes or elements. Use the **npb** and **epb** commands (referred to as time history blocks), respectively, to identify areas of the mesh requiring a more detailed accumulation of data by DYNA3D.

II. DYN3D Output Example

The following example was provided by Dr. Richard J. Fields at National Institute of Standards and Technology of the United States Department of Commerce. It has been modified from a model developed for another code.



This model forms a pair of clamps and a rectangular block of metal to be drawn as the clamps move apart. The first part forms both the rectangular metal to be drawn as well as the majority of the clamps. This removes the need of merging the nodes of the clamp with the nodes of the drawn material. The wedge portion is made as a separate part to take advantage of the transitional block boundary. The clamps are made rigid. The entire bottom face is constrained in the z-direction. Displacements are applied to the end faces of the clamps.

```
title Metal Drawing Process using DYN3D by NIST Metallurgy Div.  
  
c choose the output format  
dyna3d  
  
c set the termination time and the plot interval  
dynaopts term 1.e-3 plti 1.e-4 ; ;  
  
c load curve  
lcd 1 0 0 1 1000.;
```

```

c elastic material for the sample material being drawn
dynamats 1 24 rho 7.e-4 mhead workpiece
    e 3.e7 pr 0.3 sigy 75000 et 1.e5 efp 0.2 ;

c rigid material for the clamps
dynamats 2 20 rho 7.e-4 e 3.e7 pr 0.3 ;

c main part with both elastic and rigid materials
block 1 13 17 31 35 47;1 5 9 13;1 25;
    1 13 17 31 35 47;1 5 9 13;1 25;

c remove some of the unneeded regions between the clamps
dei 2 5; 1 2 0 3 4;

c set the default material to elastic
mate 1

c set the clamp regions of the part to rigid material
mti 1 3 0 4 6; 1 2 0 3 4; ; 2

c position some of the key nodes
mbi -2; -1 0 -4; 1 2;x -1
mbi -5; -1 0 -4; 1 2;x 1

c save the interface for the second part
bb 2 2 1 3 2 2 1;

c nodal constraints
bi ;; -1;dz 1;

c displacements
fdi -6;1 2 0 3 4;1 2;1 1 1 0 0
fdi -1;1 2 0 3 4;1 2;1 1 -1 0 0

endpart

c part to form the transitional region of the clamps
block 1 3;1 5;1 25; 13 17 1 5 1 25

c all rigid material
mate 2

c position some of the nodes
mb 1 1 1 1 1 2 x -1
mb 2 1 1 2 1 2 x -4

```

```
c transitional interface
trbb 1 2 1 2 2 2 1;

c replicate this part
lct 3 rzx my 14;ryz mx 48;rzx ryz my 14 mx 48;
lrep 0:3;

c nodal constraints
bi ; -1;dz 1;

endpart

c enter the merge phase to write the output file
merge

c merge the nodes at the interfaces
stp .001

c write the output file
write
```


III. DYN3D Output Reference

The commands found here are provided to the user so that a complete input file can be generated by **TrueGrid®**. This manual does not try to explain the meaning of these parameters. For this, the user is referred to the DYNA3D User Manual.

Command Syntax Conventions

When an arbitrarily long list of arguments are required, a semi-colon terminates the list. Sometimes the abbreviation `#_things` is used to mean “number of things”. Each command is described by an entry like the following:

command	summary description
command <i>arguments</i>	brief description of functionality with brief descriptions of what the <i>arguments</i> should be. indentation is used to indicate a list of options to the <i>arguments</i>

Remarks

When present, the Remarks section describes the command in even greater detail. It may describe the context in which the command is normally used, and other commands used in association with this command. It may describe side effects. It may describe other, similar commands. In many cases, it includes a description of where to find the command in the menus.

Example

When present, this shows the exact use of the command. If you use the dialogues, this command will be generated by simple selection options with the mouse and entering data where indicated. The command, as shown here, will appear in the session file for later reuse and possible modification. You can also enter the command into the text window or insert it into a command file to be run in batch mode.

bsd global beam cross section definition

bsd *option_list* ;
where an *option* can be:

for the Hughes-Liu beam with constant thickness
s_{thi} thickness s-thickness at both ends

tthi *thickness*

t-thickness at both ends

for the Hughes-Liu beam with variable thickness

sthi1 *thickness*

s-thickness at beginning

sthi2 *thickness*

s-thickness at ending

tthi1 *thickness*

t-thickness at beginning

tthi2 *thickness*

t-thickness at ending

for the Belytschko-Schwer beam

carea *area*

cross section area

iss *iss*

area moment of inertia about s-axis

itt *itt*

area moment of inertia about t-axis

irr *irr*

area moment of inertia about r-axis

sarea *area*

shear area of cross section

for the truss

carea *area*

cross section area

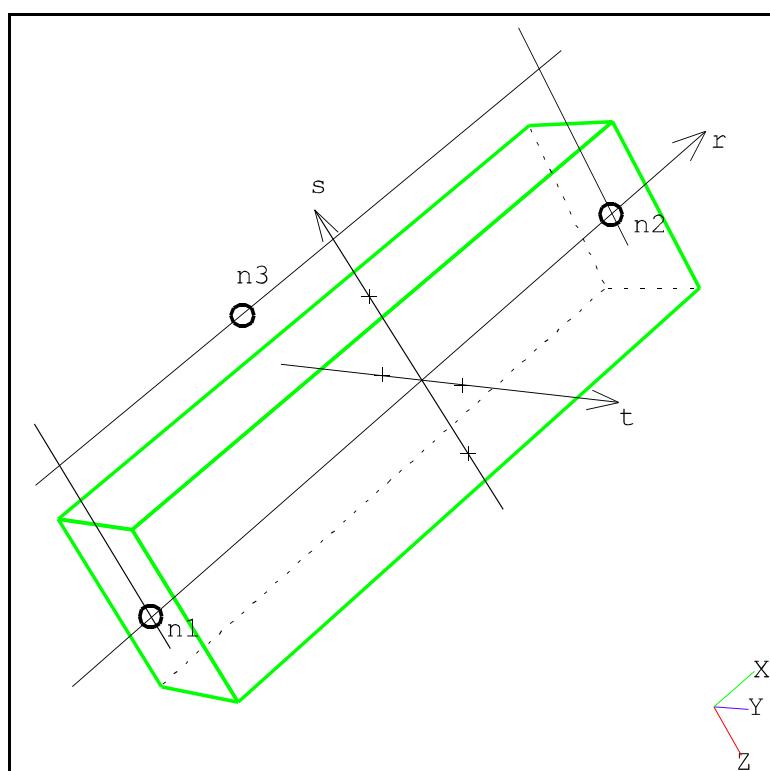


Figure 3 Beam Local Coordinate System for DYNA3D

Remarks

A third node is always required but is not significant for a truss. For other beam types, the third node is used to define the cross section orientation.

There are other options to this command, but they are not shown here because they do not apply to the DYNA3D output. For a complete description of the **bsd** command, see the **TrueGrid®** User Manual.

Since there are default cross section properties provided in the definition of the material using the **dynamats** command, not all of the parameters need to be assigned through the **bsd** command. Only use the parameters appropriate for the beam type selected in the **dynamats** command. Inappropriate parameters will be ignored.

Each cross section definition is assigned a number by you so that you can reference it when defining a set of beams with the **ibm**, **ibmi**, **jbm**, **jbmi**, **kbm**, **kbmi**, **bm**, and **beam** commands.

Example

```
bsd 2 sthi .03 tthi .03 ; ;
```

sid sliding interface definition

sid *slide_# option_list* ;

where an *option* can be

tied	tied sliding surface
sl	sliding only
sv	sliding with voids
single	single sided slide surface
dni	discrete nodes impacting surface
dnt	discrete nodes tied to surface
sets	hell element edge tied to shell element surface
nsw	nodes spot welded
break	tie-break interface
owsv	one way sliding with voids
dummy	is only used to insure that nodes in this interface will not be merged
sand type	Slide Surface with Adaptive New Definitions

where *type* can be

sms *slave_material_list* ;

mms *master_material_list* ;

auto	for automatic contact
rebar type	to define properties of REBAR 1D sliding interface where type can be any of the following:
rbrad	radius
rbstr	strength
rbshr	modulus
rbumax	displacement
rbexp	exponent
rbibond	non-negative_number
fric friction_factor	for static coefficient of friction
kfric kinetic_coefficient_of_friction	for kinetic coefficient of friction
decay exponential_decay_coefficient	for exponential decay coefficient
pen	for small penetration flag
sfif	for slave to be printed in force file
mfif	for master to be printed in force file
pnlt_s slave_penalty_factor	for slave penalty factor
pnlt_m master_penalty_factor	for master penalty factor
pnlt penalty_factor	for sliding penalty

Remarks

Sliding interfaces or contact surfaces are constructed in 3 steps. These steps can be done in any order.

1. define the properties
2. select the slave side
3. select the master side, if applicable

The **sid** command is used to define the properties. The **si** and **sii** commands are used in the part phase or the merge phase to select the nodes or faces that form the master and slave sides of the interface.

When nodes are merged, nodes across a sliding interface will not be merged. When a merge command is first issued in the merge phase, a table is written listing the number of nodes and faces associated with each sliding interface.

The **dummy** type interface is actually used to avoid merging of nodes. A sliding interface of this type is not written to the output file.

The nodes and faces of a sliding interface or contact surface can be viewed in the merge phase using the **si** option of the **co** command.

If the output option has been selected prior to using the dialogue box to make a selection, only the options available to that output option will be displayed in the dialogue box.

si select nodes or faces for sliding interface in merge phase

si *type interface_# boundary parameters* ;

where *type* and *parameters* can be one of:

n <i>node_number</i>	to select a single node
rt <i>x y z</i>	to select a node close to a Cartesian point
cy <i>rho theta z</i>	to select a node close to a cylindrical point
sp <i>rho theta phi</i>	to select a node close to a spherical point
nset <i>name_of_set</i>	to select an entire node set
fset <i>face_set</i>	to select a face set

where *boundary* can be one of

m	master side of the interface
s	slave side of the interface

si assign sliding interface to region in part phase

si *region sliding_# type options*

where

<i>sliding_#</i>	reference number for the interface
<i>type</i>	m for master and s for slave
<i>options</i>	this depends on the <i>type</i> .

sii assign sliding interfaces to progression in part phase

sii *progression sliding_# type options*

where

<i>sliding_#</i>	sliding interface reference number
<i>type</i>	m for master and s for slave
<i>options</i>	this depends on the <i>type</i> .

Example

A model was created by the use of the following command file. Some normals are displayed as circular arcs with arrows. This is caused by the normals pointing almost orthogonally from the screen. There is an angle parameter in the **co** command setting the range of such behavior. You can modify it, or totally disable it.

```
c Sliding Interface -> master side
c Part definition -> shells.
```

```

block 1 3 5 7 9;-1;1 3 5 7 9; 0 2 4 6 8; 9 ; 0 2 4 6 8;
c Definition of orientation point in Cartesian coord x,y, and z
orpt - 9 0 9
c Definition of the type of the sliding interface
sid 1 sv ;
c Assignment of region (;-1;;) and type(1 m)of slid. interf.
sii ;-1;; 1 m
c Definition of 3 global transformations around y-axis
gct 3 ry 90; ry 180; ry 270 ; c for 90,180 and 270 degrees.

c Global replication 3 times by rotation for
grep 0 1 2 3; 90,180 and 270 degrees
c Cylinder part -> tube made from hexahedrons.
cylinder 1 6; 1 3 5 7 9 11 13 15 17 19 21 23 25; 1 10;
2 4;0 30 60 90 120 150 180 210 240 270 300 330 360;0 20;
c Definition of the orientation point in the default coordinate
c system of the part (in Cylindrical coordinates r,eta,z).
orpt - 0 20 5
c Assignment of region (-2;;) and type(1 s) of slid. interf.
sii -2;; 1 s
c Assignment of region (;;-1;) and type(1 s) of slid. interf.
sii ;;-1; 1 s
c Assignment of region (;;-1;) and type(1 s) of slid. interf.
sii ;;-2; 1 s
lct 1 my 20 ; c Definition of the local transformation
lrep 1; c Transformation 1 is applied.
merge
labels size 3 c Scale the size of arrows.
rx 20 ry 20 rz 20 c Rotate mesh in window.
center c Center picture in window.
set tv disp c Set hide display option.
co si 1 m; c Display of master side of sliding interface 1
co si 1 s; c Display of slave side of sliding interface 1

```

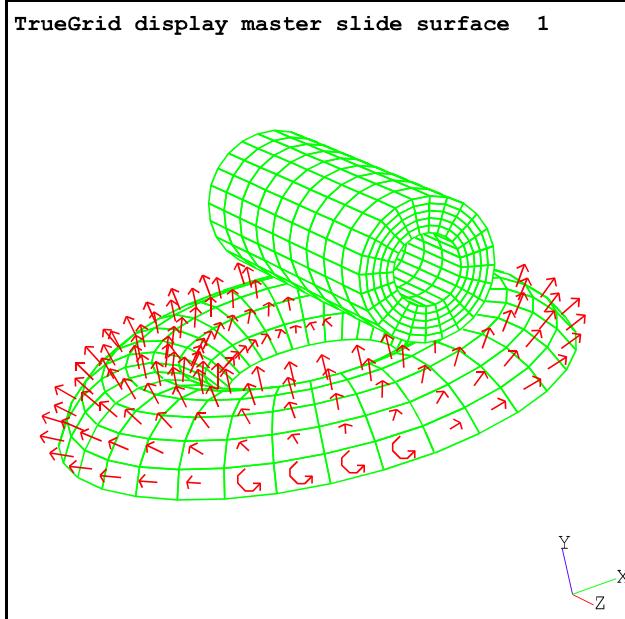


Figure 4 master side of interface

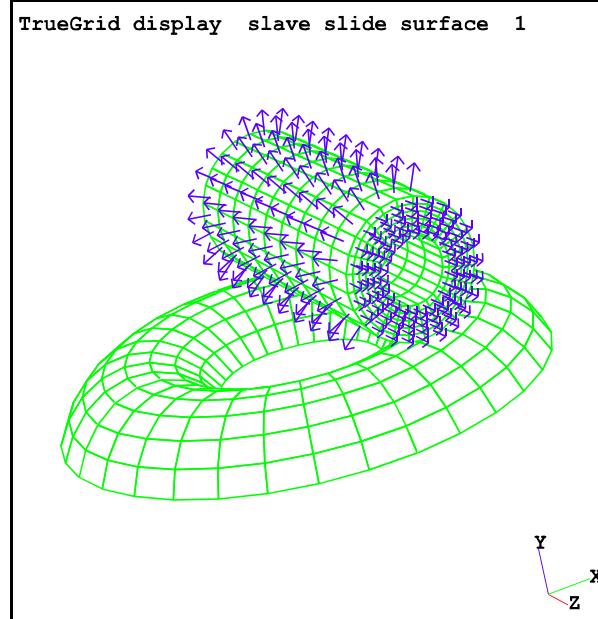


Figure 5 slave side of interface

Remarks

The *options* for the **si** and **sii** commands are not used for DYNA3D.

spd define the properties of a set of springs or dampers

spd *spring/damper_# type parameters*

where *type* is the spring or damper's material model and

parameters is one of the following:

le <i>stiffness</i>	linear elastic
lv <i>damping</i>	linear viscous
iep <i>elastic tangent yield</i>	isotropic elastic
ne <i>ld_curve_#</i>	nonlinear elastic
nv <i>ld_curve_#</i>	nonlinear viscous
nesf <i>ld_curve_# force_curve_#</i>	nonlinear elastic w/ force load curve
gn <i>loading_# unloading_# hardening tension compression</i>	general nonlinear

Remarks

A spring or damper is defined using either the **spdp** command forming a set of springs/dampers between two surfaces, or using the **spring** command to create a single spring at a time. In each case, the definition of a spring includes a reference to a material definition **spd** number.

DYNA3D, use linear elastic, linear viscous (damper), isotropic elastoplastic, nonlinear elastic, nonlinear viscous, general tabulated nonlinear, and dashpot.

If the output option has been selected prior to using the dialogue box to make a selection, only the options available to that output option will be displayed in the dialogue box.

dynaeos **DYNA3D equation of state**

dynaeos *material_# eos_type parameters_list* ;

where the *eos_type* can be

1	linear polynomial
2	JWL
3	Sack
4	Gruneisen
5	ratio of polynomials
6	linear polynomial with energy deposition
7	ignition and growth of reaction in HE
8	tabulated model with compaction
9	tabulated
11	pore collapse

where the *parameters_list* for EOS linear polynomial (type 1) is:

- c0** *constant*
- c1** *coefficient*
- c2** *coefficient*
- c3** *coefficient*
- c4** *coefficient*
- c5** *coefficient*
- c6** *coefficient*
- e0** *energy*
- v0** *volume*

where the *parameters_list* for EOS JWL (type 2) is:

- a** *constant*
- b** *constant*

r1 constant
r2 constant
omega constant
e0 energy
v0 volume

where the *parameters_list* for EOS Sack (type 3) is:

a1 constant
a2 constant
a3 constant
b1 constant
b2 constant
b0 constant
v0 constant

where the *parameters_list* for EOS Gruneisen (type 4) is:

vci intercept
s1 coefficient
s2 coefficient
s3 coefficient
gamma coefficient
sa coefficient
b0 energy
v0 volume

where the *parameters_list* for EOS ratio of polynomials (type 5) is:

a10 constant
a11 constant
a12 constant
a13 constant
a20 constant
a21 constant
a22 constant
a23 constant
a30 constant
a31 constant
a32 constant
a33 constant
a40 constant
a41 constant
a42 constant
a43 constant

a50 constant
a51 constant
a52 constant
a53 constant
a60 constant
a61 constant
a62 constant
a63 constant
a70 constant
a71 *constant*
a72 *constant*
a73 *constant*
alpha *constant*
beta *constant*
a14 *constant*
a24 *constant*
e0 *energy*
v0 *volume*

where the *parameters_list* for EOS linear polynomial with energy deposition (type 6) is:

c0 *constant*
c1 *coefficient*
c2 *coefficient*
c3 *coefficient*
c4 *coefficient*
c5 *coefficient*
c6 *coefficient*
e0 *energy*
v0 *volume*
lc *load_curve*

where the *parameters_list* for EOS ignition and growth of reaction in HE (type 7) is:

ap *constant*
bp *constant*
r1p *constant*
r2p *constant*
g *coefficient*
wpcp *constant*
ae *constant*
be *constant*
wece *constant*
r1e *constant*

r2e *constant*
fcrf *fraction*
i *coefficient*
h *coefficient*
z *exponent*
x *exponent*
y *exponent*
cp *heat_capacity*
ce *heat_capacity*
m *exponent*
e0 *energy*
t0 *temperature*

where the *parameters_list* for EOS tabulated model with compaction (type 8) is::

eps *list_strains* ;
pc *list_constants* ;
t *list_temperatures* ;
ku *list_modulus* ;*compression*
gamma *gamma*
e0 *energy*
v0 *volume*

where the *parameters_list* for EOS tabulated (type 9) is:

eps *list_strains* ;
pc *list_constants* ;
t *list_temperatures* ;
gamma *gamma*
e0 *energy*
v0 *volume*

where the *parameters_list* for EOS pore collapse (type 11) is:

mu1 *compression*
mu2 *compression*
e0 *energy*
mu0 *compression*
virgin *load_curve_pairs* ;
crushed *load_curve_pairs* ;

NOTE: The following EOS models are under development

where the *parameters_list* for EOS Ignition and Growth of Reaction in HE 3-Term (type 13) is:

where the *integration_rule_#* can be

n	positive for the number of points using the trapezoidal rule
0	Gauss
-n	negative of the user specified rule number (from sind command)

shth *thickness*

shth1 *thickness*

shth2 *thickness*

shth3 *thickness*

shth4 *thickness*

shloc *location*

where *location* can be

1	top surface
0	middle surface
-1	bottom surface

beam *features_list*

with the following *features*

elfom *option*

where the *option* can be

hl	Hughes-Lui beams
bt	Belytschko-Tsay beams
truss	

shear factor

quad *option*

where the *option* can be

1	for a truss
2	for 2x2 Gauss quadrature
3	for 3x3 Gauss quadrature
4	for 3x3 Lobatto integration
5	for 4x4 Gauss quadrature

bmcross *shape*

where the *shape* can be

0	for rectangular
1	for tubular

sthi *thickness*

tthi *thickness*

sthi1 *thickness*

sthi2 *thickness*

tthi1 *thickness*

tthi2 *thickness*

sloc *location*

where *location* can be

1	meaning the side where s is 1
0	meaning centered
-1	meaning the side where s is -1

tloc *location*

where *location* can be

1	meaning the side where t is 1
0	meaning centered
-1	meaning the side where t is -1

tshell *features_list*

with the following *features*

shear *shear*

tsti *#_points*

quad *integration_rule_#*

rho *density*

where the *properties_list* is specific to the selected material type:

Experimental Material Model - Material type 0

pij *value* repeat as often as is needed

where *i* can be from 3 to 8 (index to the record number of the material definition)

where *j* can be from 1 to 8 (index to the field number of the material definition)

Elastic - Material type 1

e *modulus*

pr *ratio*

Orthotropic Elastic - Material type 2

ea *ea*

eb *eb*

ec *ec*

prba *vba*

prca *vca*

prcb *vcb*

gab *gab*

gbc *gbc*

gca *gca*

aopt *option parameters* for material orientation

where the *option* can be one of

0	for by nodes
1	for by point and element center
2	for by normal vectors
3	for by cross product with shell normal (shell elements only)

where the *parameters* can be

xp <i>x-coordinate</i>	for aopt 1
yp <i>y-coordinate</i>	for aopt 1
zp <i>z-coordinate</i>	for aopt 1
ax <i>x-component</i>	for aopt 2
ay <i>y-component</i>	for aopt 2
az <i>z-component</i>	for aopt 2
dx <i>x-component</i>	for aopt 2
dy <i>y-component</i>	for aopt 2
dz <i>z-component</i>	for aopt 2
vx <i>x-component</i>	for aopt 3
vy <i>y-component</i>	for aopt 3
vz <i>z-component</i>	for aopt 3
beta <i>angle</i>	

Kinematic/Isotropic Elastic-Plastic - Material type 3

e *modulus*
pr *ratio*
sigy *stress*
etan *modulus*
beta *parameter*

Thermo-Elastic-Plastic - Material type 4

temp *temperature_list* ;
e *modulus_list* ;
pr *ratio_list* ;
alpha *secant_list* ;
sigy *stress_list* ;
etan *modulus_list* ;

Soil And Crushable Foam - Material type 5

g *modulus*
ku *modulus*
a0 *yield*
a1 *yield*
a2 *yield*
pc *pressure*
vs *strain_list* ;
p *pressure_list* ;

Viscoelastic - Material type 6

k *modulus*
g0 *modulus*

gi modulus

beta constant

mflag option

where the *option* can be

0

which means that beta is the delay constant

1

which means that beta is the time relaxation constant

Blatz-Ko Hyperelastic Rubber - Material type 7

g modulus

High Explosive Burn - Material type 8

d velocity

pcj pressure

Fluid - Material type 9

pc pressure

mu coefficient

Isotropic-Elastic-Plastic-Hydrodynamic - Material type 10

g modulus

sigy stress

etan modulus

pc pressure

a1 coefficient

a2 coefficient

ispall model

where the *model* can be

pl

for pressure limit

max

for maximum principal stress spall criterion

hydro

for hydrostatic tension spall criterion

eps strain_list ;

up to 16 values

es stress_list ;

up to 16 values

Steinberg-Guinan High Rate Elastic-Plastic - Material type 11

g0 modulus

sig0 stress

beta constant

n exponent

gama strain

sigm stress

b modulus

bpm stress

h coefficient
f exponent
t0 temperature
gam0 gamma
sa constant
pc pressure
ispall model

where the *model* can be

pl	for pressure limit
max	for maximum principal stress spall criterion
hydro	for hydrostatic tension spall criterion

a atomic_weight
r r_prime
spall
ivar option

where the *option* can be

0	for cold compression polynomial coefficient in eta
1	for cold compression polynomial coefficient in mu

ec0 coefficient
ec1 coefficient
ec2 coefficient
ec3 coefficient
ec4 coefficient
ec5 coefficient
ec6 coefficient
ec7 coefficient
ec8 coefficient
ec9 coefficient

Isotropic-Elastic-Plastic - Material type 12

g modulus
sigy stress
eh modulus
k modulus

Elastic-Plastic With Failure - Material type 13

g modulus
sigy stress
eh modulus
fs strain
fp pressure
k modulus

Soil And Crushable Foam With Failure - Material type 14

g modulus

ku modulus

a0 constant

a1 constant

a2 constant

pf pressure

iflag flag

where the *flag* can be

0

for hydrostatic tension

1

for maximum principal stress

sigmaf stress

vs strain_list ;

ps pressure_lis ;

Johnson/Cook Elastic-Plastic - Material type 15

g modulus

a stress

b coefficient

n exponent

sc coefficient

m exponent

tm temperature

tr temperature

x0 rate

sh heat

ispall model

where the *model* can be

pl

for pressure limit

max

for maximum principal stress spall criterion

hydro

for hydrostatic tension spall criterion

iter flag

where the *flag* can be

0

for fast approximate solution

1

for accurate iterative solution

d1 parameter

d2 parameter

d3 parameter

d4 parameter

d5 parameter

e modulus

pr ratio

dterit *step_size*

Concrete/Geological Model - Material type 16

pr ratio
g modulus
sigy stress
a0 cohesion
a1 coefficient
a2 coefficient
b1 factor
a0f cohesion
a1f coefficient
r percent
emr modulus
prr ratio
sigma0 stress
tm modulus
lc load_curve
lcr load_curve
eps list_strain ;
es list_stress ;
p list_pressure ;

Isotropic Elastic-Plastic With Oriented Crack - Material type 17

e modulus
pr ratio
sigy stress
eh modulus
fs strength
pc pressure

Power Law Isotropic Elastic-Plastic - Material type 18

e modulus
pr ratio
k coefficient
n exponent

Strain Rate Dependent Isotropic Elastic-Plastic - Material type 19

e modulus
pr ratio
lcs0 load_curve
etan modulus

lce *load_curve*
lce *load_curve*
lcfs *load_curve*
tss *step_size*

Rigid - Material type 20

e *modulus*

pr *ratio*

bpm *options* ;

where an *option* can be

dof *flag*

where *flag* can be

1	x-translational degree-of-freedom
2	y-translational degree-of-freedom
3	z-translational degree-of-freedom
4	translational motion in the given vector direction (use v below)
5	x-rotational degree-of-freedom
6	y-rotational degree-of-freedom
7	z-rotational degree-of-freedom
8	rotational motion about the given vector (use v below)

lcid *load_curve_#*

sf *scale_factor*

v *x₀ y₀ z₀*

rbv *load_curve amplitude fx fy fz* (obsolete)

Thermal Orthotropic Elastic - Material type 21

ea *modulus*

eb *modulus*

ec *modulus*

prba *ratio*

prca *ratio*

prcb *ratio*

alpa *coefficient*

alpb *coefficient*

alpc *coefficient*

gab *modulus*

gbc *modulus*

gca *modulus*

aopt *option parameters*

for material orientation

where the *option* can be one of

0 for by nodes

1	for by point and element center
2	for by normal vectors
3	for by cross product with shell normal (shell elements only)

where parameters can be

xp <i>x</i> -coordinate	for aopt 1
yp <i>y</i> -coordinate	for aopt 1
zp <i>z</i> -coordinate	for aopt 1
ax <i>x</i> -component	for aopt 2
ay <i>y</i> -component	for aopt 2
az <i>z</i> -component	for aopt 2
dx <i>x</i> -component	for aopt 2
dy <i>y</i> -component	for aopt 2
dz <i>z</i> -component	for aopt 2
vx <i>x</i> -component	for aopt 3
vy <i>y</i> -component	for aopt 3
vz <i>z</i> -component	for aopt 3
beta angle	

Fiber Composite With Damage - Material type 22

ro density	
ea modulus	
eb modulus	
ec modulus	
k modulus	
sn strength	
syz strength	
szx strength	
prba ratio	
prcb ratio	
prca ratio	
gab modulus	
gbc modulus	
gca modulus	
aopt option parameters	for material orientation
where the <i>option</i> can be one of	
0	for by nodes
1	for by point and element center
2	for by normal vectors
3	for by cross product with shell normal (shell elements only)
where the <i>parameters</i> can be	
xp <i>x</i> -coordinate	for aopt 1

yp <i>y-coordinate</i>	for aopt 1
zp <i>z-coordinate</i>	for aopt 1
ax <i>x-component</i>	for aopt 2
ay <i>y-component</i>	for aopt 2
az <i>z-component</i>	for aopt 2
dx <i>x-component</i>	for aopt 2
dy <i>y-component</i>	for aopt 2
dz <i>z-component</i>	for aopt 2
vx <i>x-component</i>	for aopt 3
vy <i>y-component</i>	for aopt 3
vz <i>z-component</i>	for aopt 3

axes *flag*
 where the *flag* can be

1	for the default
2	for switch material axes a and b
3	for switch material axes a and c

sc *strength*
xt *strength*
yt *strength*
yc *strength*
alpha *parameter*
beta *list_angles* ;

Thermal Orthotropic Elastic With Variable Properties - Material type 23

ea <i>ea_list</i> ;	
eb <i>eb_list</i> ;	
ec <i>ec_list</i> ;	
vba <i>vba_list</i> ;	
vca <i>vca_list</i> ;	
vcb <i>vcb_list</i> ;	
aa <i>aa_list</i> ;	
ab <i>ab_list</i> ;	
ac <i>ac_list</i> ;	
gab <i>gab_list</i> ;	
gbc <i>gbc_list</i> ;	
gca <i>gca_list</i> ;	
t <i>temperature_list</i> ;	
angles <i>angle_list</i> ;	
aopt <i>option parameters</i>	for material orientation

where the *option* can be one of

0	for by nodes
1	for by point and element center

2	for by normal vectors
3	for by cross product with shell normal (shell elements only)

where the *parameters* can be

xp <i>x</i> -coordinate	for aopt 1
yp <i>y</i> -coordinate	for aopt 1
zp <i>z</i> -coordinate	for aopt 1
ax <i>x</i> -component	for aopt 2
ay <i>y</i> -component	for aopt 2
az <i>z</i> -component	for aopt 2
dx <i>x</i> -component	for aopt 2
dy <i>y</i> -component	for aopt 2
dz <i>z</i> -component	for aopt 2
vx <i>x</i> -component	for aopt 3
vy <i>y</i> -component	for aopt 3
vz <i>z</i> -component	for aopt 3

Rate-Dependent Tabular Isotropic Elastic-Plastic - Material type 24

e *modulus*
pr *ratio*
sigy *stress*
et *modulus*
efp *strain*
dtcrit *time*
lc *load_curve*
eps *strain*
es *stress*

Extended Two Invariant Geologic Cap - Material type 25

k *modulus*
g *modulus*
alpha *parameters*
theta *coefficient*
gamma *coefficient*
beta *exponent*
r *ratio*
d *exponent*
w *coefficient*
x0 *parameter*
cbar *coefficient*
n *parameter*
nplot *option*

where the *options* are

1	hardening variable, k
2	cap - j1 axis intercept, x(k)
3	volumetric plastic strain
4	first stress invariant, j1
5	second stress invariant, square root of j2
8	response mode number
9	number of iterations

ltype *option*

where the *options* are

1	soil or concrete (cap surface may contract)
2	rock (cap surface does not contract)

ivec *option*

where the *options* are

0	vectorization (fixed number of iterations)
1	fully iterative

t cutoff

Metallic Honeycomb - Material type 26

e modulus

pr ratio

sigy stress

sigaa load_curve

sigbb load_curve

sigcc load_curve

ssrv load_curve

crv volume

ea modulus

eb modulus

ec modulus

gab modulus

gbc modulus

gca modulus

Compressible Mooney-Rivlin Hyperelastic Rubber - Material type 27

a coefficient

b coefficient

pr ratio

Resultant Plasticity - Material type 28

e modulus

pr ratio

sigy stress

etan modulus

Closed-Form Update Elastic-Plastic For Shells - Material type 30

e modulus

pr ratio

sigy stress

etan modulus

Frazer-Nash Hyperelastic Rubber - Material type 31

g001 coefficient

g010 coefficient

g020 coefficient

g100 coefficient

g101 coefficient

g110 coefficient

g200 coefficient

g210 coefficient

g300 coefficient

g400 coefficient

ilimit option

where the *option* can be

0 to stop if strain limits are exceeded

1 to continue if strain limits are exceeded

stmx strain

stmn strain

Ramberg-Osgood Elastic-Plastic - Material type 32

gammay strain

tauy stress

alpha coefficient

r exponent

k modulus

General Anisotropic Elastic-Plastic - Material type 33

ea modulus

eb modulus

ec modulus

r coefficient

acp coefficient

qbc coefficient

qab coefficient

qac coefficient

prba *ratio*

prca *ratio*

prcb *ratio*

aopt *option parameters* for material orientation

where the *option* can be one of

0	for by nodes
1	for by point and element center
2	for by normal vectors
3	for by cross product with shell normal (shell elements only)

where the *parameters* can be

xp <i>x-coordinate</i>	for aopt 1
yp <i>y-coordinate</i>	for aopt 1
zp <i>z-coordinate</i>	for aopt 1
ax <i>x-component</i>	for aopt 2
ay <i>y-component</i>	for aopt 2
az <i>z-component</i>	for aopt 2
dx <i>x-component</i>	for aopt 2
dy <i>y-component</i>	for aopt 2
dz <i>z-component</i>	for aopt 2
vx <i>x-component</i>	for aopt 3
vy <i>y-component</i>	for aopt 3
vz <i>z-component</i>	for aopt 3

sigya *stress*

beta *angle*

eap *modulus*

gbc *modulus*

gab *modulus*

gac *modulus*

npss *substeps*

epsap *list_strain* ;

sigmaya *list_stress* ;

Normal Anisotropic Elastic-Plastic For Shells - Material type 34

e *modulus*

pr *ratio*

sigy *stress*

etan *modulus*

r *parameter*

Elastic-Plastic With Forming Limit Diagram - Material type 35

e *modulus*

ifld *option*

where the *option* can be

- 1** for total
- 2** for incremental
- 3** for damage

pr ratio
lclh load_curve
lcrh load_curve
lcpx load_curve
lcedf load_curve
lcedm load_curve
sigy stress
scldev factor
etan modulus
beta parameter
ep list_strain ;
sigmay list_stress ;

Brittle Damage Model - Material type 36

e modulus
pr ratio
ft0 limit
fs0 limit
gc toughness
beta retention
nu viscosity

NOTE: The following material models are under development.

Three-Invariant Viscoplastic Cap Model - Material type 37

Bammann Plasticity Model - Material type 38

Snadia Damage Model - Material type 39

Fahrenthold Brittle Damage - Material type 40

Fabric with Damage - Material type 41

Multi-Material Shell Element Model - Material type 42

Transversely Isotropic Visco-Hyperelasticity - Material type 43

Rigid Foam - Material type 44

DTRA Concrete/Geological - Material type 45

Anisotropic Elastic - Material type 46

MIG - Material Interface Guide - Material type 47

Visco-Elastic with Statistical Crack Mechanics - Material 48

LANL Hyperfoam - Material type 49

Braided Composite Model with Damage - Material type 50

Uni-Directional Elasto-Plastic Composite - Material type 56

Uni-Directional Elasto-Plastic Composite - Material 62

Visco-Hyper Elastic - Material type 63

Steinberg-Guinan with 3-D Failure - Material 64

K&C Concrete/Geological - Material type 65

Brittle Damage Model with Power-law Plasticity - Material 70

Delamination Element Material Model

Cohesive Element Material Model

Remarks

You must be in the control phase to issue this command. When you first start **TrueGrid®** you are in the control phase and it is convenient to issue this command at that time. However, one can return to the control phase anytime using the **control** command.

Example

```
para
c sheet metal
e_stl      200.0e3      c modulus of elasticity for steel
v_stl      0.33         c poisson ratio for steel
yld_stl    207          c yield stress for steel
```

```

tan_stl      200      c tangent modulus for steel
den_stl      7.86e-9 ; c density of steel
dynamats 12 3 shell
  e %e_stl pr %v_stl sigy %yld_stl etan %tan_stl
  beta 0. rho %den_stl tsti 3;
dynamats 18 3 beam
  e %e_stl pr %v_stl rho %den_stl sigy %yld_stl
  etan %tan_stl beta 0.
  bmcross 1 elfom hl sthi 25 tthi 10 quad 3 sloc 0  tloc 0 ;
dynamats 9 1 e 1 pr 0.35 ;

```

dynaopts **DYNA3D analysis options**

dynaopts *options*

where any or all of the following *options* can be invoked:

iif *interval*

stsm *factor*

time interval between writes of the interface segment save file
 minimum time step size for thin shell element using the
 materials kinematic/isotropic elastic-plastic, strain rate
 dependent elastic-plastic, or rate dependent tabular isotropic
 elastic-plastic

set the reaction force print flag

rfpf *flag*

where the *flag* can be

0

1

no printing

reactions are printed

defpf *flag*

where the *flag* can be

0

1

no printing

forces in all elements are written

edsdf *flag*

where the *flag* can be

0

1

failed elements not deleted

2

failed elements deleted

10

triggers a debug run

11

same as 0 except failed shells not printed

12

same as 1 except failed shells not printed

same as 2 except failed shells not printed

gvst *acceleration direction mat_1 mat_2 ... ;* initialize gravity stress

density_1 depth_1 density_2 depth_2 ... ;

stat *interval*

number of steps between problem status reports (default
 1000)

yldb *yield*

brode function yield in ktons

hiteb *distance*

brode function height of burst

xb0 <i>x</i>	model x-coordinate of brode origin
yb0 <i>y</i>	model y-coordinate of brode origin
zb0 <i>z</i>	model z-coordinate of brode origin
tb0 <i>time</i>	model time of brode time origin
lcb1 <i>load_curve</i>	brode function arrival versus range load curve relative to brode origin
lcb2 <i>load_curve</i>	brode function yield scaling versus time load curve relative to brode
clb <i>factor</i>	brode function conversion factor kft to DYNA3D length units
ctb <i>factor</i>	brode function conversion factor milliseconds to DYNA3D time units
cpb <i>factor</i>	brode function conversion factor psi to DYNA3D pressure units
ticsf <i>interval</i>	time interval between output cross section forces
ngrav <i>x_acceleration</i> <i>load_curve</i>	prescribed base acceleration
<i>y_acceleration</i> <i>load_curve</i>	
<i>z_acceleration</i> <i>load_curve</i>	
xvel <i>x_velocity</i> <i>load_curve</i>	angular velocity about the x-axis
yvel <i>y_velocity</i> <i>load_curve</i>	angular velocity about the y-axis
zvel <i>z_velocity</i> <i>load_curve</i>	angular velocity about the z-axis
term <i>time</i>	termination time
prt1 <i>interval</i>	time interval between writes of time history node and element print block plot data
plti <i>interval</i>	time interval between writes of state plot database for all nodes and elements and, optionally, the interface force database containing pressures and shear traction for all sliding interfaces
nrest <i>time_step</i>	number of time steps between writes of the saved restart files
nrunr <i>time_step</i>	number of time steps between writes of the continuously overwritten running restart file
itss <i>time_step</i>	initial time step size
pnlt <i>factor</i>	global scale factor for sliding interface penalty stiffness controlling inter-penetration and stability (default 0.1)
teo <i>load_curve</i>	thermal effects option
where a non-positive value for <i>load_curve</i> means	
0	no thermal effects
-1	nodal temperatures in TOPAZ3D generated plot files
-2	nodal temperatures use temperature option 2
-3	TOPAZ3D coupling
-4	nodal temperatures use temperature option2; subsequent temperature history are defined in TOPAZ3D generated plot files.

-9999	nodal temperatures use temperature option 1
tssf <i>factor</i>	time step scale factor (default is 0.67 for high explosives, 0.90 otherwise)
lcmax <i>load_curve</i>	load curve number that limits maximum time step size (optional)
ssdm	write shell strain tensor at inner and outer surface
snrs <i>option</i>	Hughes-Liu shell normal update option
where <i>option</i> is the number of time steps between computations or	
-2	unique nodal fibers
-1	compute normals each time step
1	compute on restarts
<i>n</i>	compute every <i>n</i> steps
stup	shell thickness change due to membrane straining
sfor <i>option</i>	default shell element formulation
where the <i>option</i> can be	
hl	Hughes-Liu
bt	Belytschko-Tsay
bciz	triangle
c0	triangle
membrane	
yase	YASE with full in-plane integration
yase2	Bath-Dvorkin (full integration)
bd	Belytschko-Lin-Tsay with selective-reduced integration
blt	
tsmin <i>factor</i>	reduction factor for initial time step size to determine minimum time step size. When this minimum time step is reached, DYNA 3D terminates with a restart dump.
itrx #_iterations	number of iterations between convergence checks for dynamic relaxation in quasi-static problems (default 250)
tolrx <i>tolerance</i>	convergence tolerance for dynamic relaxation option (default 0.0001)
facrx <i>factor</i>	dynamic relaxation static analysis velocity reduction factor (default 0.995)
scftrx <i>factor</i>	scale factor for computed time step during dynamic relaxation
stss <i>option</i>	alternative methods for approximating the maximum stable time step size for 4-node shell elements
where the <i>option</i> can be	
0	characteristic length is area/longest side
1	characteristic length is area/longest diagonal
2	based on bar wave, shortest side, area/longest side
plas <i>option</i>	plane stress constitutive integration algorithms for elastoplastic shell material models

where the *option* can be

1

2

3

prtflg

drdb

rayl alpha

ihq option

where the *option* can be

1

2

3

4

5

6

7

8

9

10

12

qh coefficient

q1 coefficient

q2 coefficient

iterative plasticity with 3 secant iterations

full iterative plasticity

stress scaling non-iterative plasticity

print element time step sizes on the first cycle

write the taurus database at every convergence check during dynamic relaxation

global generalized Rayleigh damping mass proportional coefficient

select an hourglass stabilization method

standard DYNA3D (viscous form)

Flanagan-Belytschko (viscous form)

Flanagan-Belytschko with exact volume integration (viscous form)

stiffness with Flanagan-Belytschko

stiffness with Flanagan-Belytschko with exact volume

bricks: selective-reduced 8-point hexahedral element (B-bar)
shells: viscous form type 2 and stiffness form type 3

bricks: physical stabilization (models 1, 40, 56 & 62 only)

total displacement physical stabilization (models 2, 7, 21, 23, 27, 31, 43, 46, 60 & 63 only)

bricks: physical stabilization - exact volume (models 1, 40, 56 & 62 only)

bricks: total displacement physical stabilization - exact volume (models 2, 7, 21, 23, 27, 31, 43, 46, 60 & 63 only)

bricks: fully integrated, 8-pt. hexahedral element

hourglass stabilization coefficient

quadratic bulk viscosity coefficient for added stability and resolution with shock waves

linear bulk viscosity coefficient for added stability and resolution with shock waves

NOTE: The following options are under development.

debug

DYNA3D places various debug information into the plot files and terminates.

ndacc

high accuracy coordinates for MILI plot data

drstep steps

maximum time steps allowed in dynamic relaxation phase. Execution ends or switches to transient phase after this step limit is reached regardless of convergence.

drtime <i>time</i>	Maximum time allowed in dynamic relaxation phase. Execution ends or switches to transient phase after this time limit is reached regardless of convergence.
hgmodes <i>modes</i>	Select the number of enhanced modes used in the physically stabilized brick element hourglass control (default=3). <i>Modes</i> can either be 3 or 6. This feature only applies to material models 3, 4, and 18 when hourglass type 7 or 9 are used.
iforce	activate the interface force output
iresett <i>flag</i>	start time designation
where <i>flag</i> can be	
0	simulation start time is set to 0. This is used for most cases.
1	simulation start time is set to the time recorded in the stress/deformation initialization file.
2	same as 1, but also read the initial time step from the stress/deformation initialization file.
mili <i>file_name</i>	family name of MILI format output files
normck <i>flag</i>	check for consistent slide surface normals
where <i>flag</i> can be	
0	slide surface normals are not checked.
1	consistency check is performed and warning is printed if inconsistent normals are detected
2	consistency check is performed and program terminates if inconsistent normals are detected
numrrf <i>n</i>	Number of running restart data dumps maintained
pressure_display	Activate the external pressure display for segments listed in the Pressure Loads section
reaction_force_out	Activate the nodal reaction forces output into the MILI plot database.
rotational_vel_out	Activate the rotational velocity/displacement output
smp_dynamic <i>smp</i>	Assign number of SMP processors used statically or dynamically
where <i>smp</i> can be	
0	Static - always use <i>threads</i> number of processors during run
1	Dynamic - use <i>threads</i> or less number of processor during run to instantaneously maximize use of system resources.
taurus_plot <i>off</i>	Disable the TAURUS state plot output.
threads <i>threads</i>	Number of shared-memory processors to use
zero_init_vol <i>vol</i>	On initialization, reset the relative volume of brick elements to unity when an element's initial volumetric strain (in absolute value) is less than <i>vol</i>
ptsming <i>ptsming</i>	Default time-step size factor for element deletion

where $ptsming$ can be

0

greater than **0**

Option inactive

Elements with non-zero $ptsming$ will be deleted when their time-step size becomes less than $ptsmin$ times the initial global time-step size. Brick elements will also be deleted when their volume becomes non-positive.

tsming $tsming$ Default time-step size for element deletion

where $tsming$ can be

0

greater than **0**

Option inactive

Elements with non-zero $tsmins$ will be deleted when their time-step size becomes less than $tsmin$. Brick elements will also be deleted when their volume becomes non-positive.

Rename the ASCII output file filename

ascii_output_file *filename*

rigid_wall *irigid*

where *irigid* can be

0

1

No forces print to file

Forces printed to file

Generate verbose input and initialization data in hsp file

verbose_hsp

penctime *file_name*

penhnose *hnose*

pen_off *off_time*

pennose *node_num*

PENCRV3D input file in a DYNA3D-PENC RV 3D analysis.

define the PENCRV3D nose height

turn off PENCRV3D when the analysis time exceeds *off_time*

Designate *node_num* as the nose tip of the projectile in a DYNA 3D PENC RV3D analysis.

fraction of the penetrator represented by the mesh

centroid of the projectile tail in a DYNA3D PENCRV3D analysis

Name of the file that contains beam element definitions

beamfile *file_name*

Name of the file that contains hex element definitions

brickfile *file_name*

Name of new input file to be read from.

include* *file_name*

Name of the file that contains auxiliary keyword definitions.

infree *file_name*

Name of the file that contains the load curve definitions

loadcurvefile *file_name*

Name of the file that contains material definitions

matfile *file_name*

Name of the stress/deformation file to be created at the end of either a regular or restart analysis.

Name of the file that contains nodal definitions

nodefile *file_name*

Name of the file that contains pressure definitions

pressurefile *file_name*

Name of the file that contains shell element definitions

shellfile *file_name*

Name of the file that contains the sliding interface definitions

slidefile *file_name*

Name of the file that contains thick shell element definitions

tshellfile *file_name*

Name of the file that contains the initial nodal velocity definitions

velofile *file_name*

ptsmin_ex <i>ptsmin mat_list</i> ;	Set <i>ptsmin</i> for all other material numbers except those in the list
ptsmin_in <i>ptsmin mat_list</i> ;	Set <i>ptsmin</i> for materials in the list
tsmin_ex <i>tsmin mat_list</i> ;	Set <i>tsmin</i> for all other material numbers except those in the list
tsmin_in <i>tsmin mat_list</i> ;	Set <i>tsmin</i> for materials in the list
verbose_hsp	Generate verbose input and initialization data in hsp file
rigid_wall <i>irigid</i>	Print all rigid-wall normal forces to FORCES file
where <i>irigid</i> can be	
0	No forces print to file (default).
1	Forces printed to file.
dump_times <i>llist_times</i> ;	write a restart file after the analysis reaches a time in the list
dt_plot <i>dt_list</i> ;	write a plotfile for each time step
fdt_plot <i>fdt_list</i> ;	write a plotfile for each factor multiplied by the time step in dt_plot
group_velocity_ex <i>mat_list</i> ;	Include in the MILI time-history and plot files the mass averaged velocity or this group of materials.
group_velocity_in <i>mat_list</i> ;	Include in the MILI time-history and plot files the mass averaged velocity for this group of materials.
mat_sv_ex <i>mat_list</i> ; <i>name_list</i> ;	Alter which material model dependent state variables and peak derived variables are included in the plotfile for all brick-element materials in the model except for the materials in the list
mat_sv_in <i>mat_list</i> ; <i>name_list</i> ;	Alter which material model dependent state variables and peak derived variables are included in the plotfile for the materials in the list
max_mises	Track the maximum value of the von Mises stress with time.
min_press	Track the minimum value of the pressure with time.
max_press	Track the maximum value of the pressure with time.
min_prin1	Track the minimum value of the 1st principle stress with time.
max_prin1	Track the maximum value of the 1st principle stress with time.
min_prin3	Track the minimum value of the 3rd principle stress with time.
max_prin3	Track the maximum value of the 3rd principle stress with time.
plotall	Select all state variables for this material model to be included in plotfile for the current material number chosen.
plotno <i>state_variable_name</i>	Exclude the variable from the plotfile for the current material chosen.
plotnone	Exclude all state variables from the plotfile for the current material chosen.

nip_thickness *num* For elements with more than one integration point, determine the number of integration points data is output for. For hex elements, the *num* is limited to 1 or the actual number of integration points.

nip_inplane *num* For multiple in-the-plane integrated shell & beam elements, output data at *num* in-plane locations. The number *num* is limit to 1 or the actual number of in-plane integration points.

maxpfile *file_name* *mat_list* ; Write the maximum pressure for the materials in the list at the time history output frequency to a text file.

pencvmat *lst* Provide a list of materials that are included in the projectile in a DYNA3D PENCRV3D analysis. Listed materials must only be associated with elements that are part of the projectile.

where *lst* can be

list *mat_list* ; List of materials must be provided.

exclude *mat_list* ; List of excluded materials must be provided.

all All materials are considered as part of the projectile.

cavity_expansion *load_curve* Initialize a new cavity expansion model and designate the pressure segments associated with the load curve as belonging to this model.

cavity0 *c0* First cavity expansion coefficient for the current layer

cavity1 *c1* Second cavity expansion coefficient for the current layer

cavity2 *c2* Third cavity expansion coefficient for the current layer

cavitycyl cylindrical cavity expansion idealization for the present C.E. model.

fscale *fscale* current layer

load_curve *lc_num* *start_time* load curve to scale the C.E. pressures.

normal *nx ny nz* normal of the target surface

pen_off *off_time* turn off this cavity expansion model when the analysis time exceeds *off_time*

pennose *node_num* tip node of the body in this cavity expansion model.

pentail *node_num* tail node of the body in this cavity expansion mode.

point_fs *px py pz* reference point lies on the upper surface of the target's top layer

sc_radius *rad* minimum spherical/cylindrical radius

spherical spherical cavity expansion idealization for the present C.E. model.

thickness *thick* thickness of the current layer

velocity_min *velmin* velocity cutoff used in calculating the C. E. pressures

velocity_trans *veltrans* transition velocity used in calculating the optional decay factor

closed_volume *law* *cutoff* *ld_curve* *pressure* *power* *plane* *node*

pressure loads associated with load curve are determined by the gas pressure in a closed volume.

where *law* can be

ideal_gas The current pressure is computed

cutoff	Negative pressure cutoff. Only positive pressure (compression) is allowed on associated element facets.
no_cutoff	Both positive and negative pressures are admissible.
where <i>plane</i> can be	
x	Project the closed volume to a plane normal to the global x-axis for volume calculation.
y	Project the closed volume to a plane normal to the global y-axis for volume calculation.
z	Project the closed volume to a plane normal to the global z-axis for volume calculation.
any	Project the closed volume to an arbitrary plane for volume calculation.
where <i>node</i> can be	
any	DYNA3D chooses the node to define the plane of projection.
n	Plane of projection passes through node n. n must be a positive integer.
dam_base <i>direction coordinate</i>	must be used if Westergaard hydrodynamic pressure is present in the Pressure Loads section
where <i>direction</i> can be	
+x	
-x	
+y	
-y	
+z	
-z	
free_surface <i>direction coordinate</i>	must be used if hydro static/hydrodynamic pressure is present in the Pressure Loads section
where <i>direction</i> can be	
+x	
-x	
+y	
-y	
+z	
-z	
gravity <i>g</i>	acceleration of gravity, <i>g</i> , for hydrostatic or hydrodynamic pressure calculation use.
hydro_density <i>rho</i>	water (or other fluid) density for hydrostatic or hydrodynamic pressure calculation use
hydro_cutoff <i>switch</i>	hydrodynamic pressure
where <i>switch</i> can be	

on	the hydrodynamic pressure will be set to zero if the instantaneous base acceleration generates negative pressure according to the Westergaard's formula
off	default
shear_traction <i>ld_curve option</i>	Designate the pressure loads associated with Load Curve to be applied in a tangential direction instead of the segment normal direction.

where the tangential direction is controlled by *option*

local_r	tangential traction is applied in direction of local r-direction
local_s	tangential traction is applied in direction of local s-direction
aux_vector <i>x y z</i>	tangential traction is applied in a direction that is determined by the segment normal vector and an auxiliary vector.
aux_nodes <i>x y z</i>	tangential traction is applied in a direction that is determined by the segment normal vector and an auxiliary vector
weibull mat	Calculate the Weibull statistic probability of failure (volume based) for material
wistrength <i>value</i>	specify the strength value as for the current Weibull material definition.
wiexponent <i>m</i>	specify the exponent value as <i>m</i> for the current Weibull material definition.
wigamma <i>offset</i>	specify the optional offset value of for the current Weibull material definition.

Remarks

This command sets parameters that are found in the control cards for the DYNA3D input file format.

You must be in the control phase to issue this command. When you first start **TrueGrid®** you are in the control phase and it is convenient to issue this command at that time. However, one can return to the control phase anytime using the **control** command.

All keywords that have one indentation from the left are keyword options to this command. Many of these keywords are followed by additional data. If the additional data is one of several options, those options are further indented. Each further indentation indicates a set of options to the keyword above the list.

For example, the **iif** option must be followed by a number that is the factor. The **sfor** option must be followed by one of **hl**, **bt**, **bciz**, **c0**, **membrane**, **yase**, **yase2**, **bd**, or **blt**.

This command can be issued to set just one option or it can continue with as many options as desired until this list of options is terminated with a semi-colon. This command can be issued as many times as desired.

Because of the complexity of this command, it is advised that you use the dialogue box to select the options you require. The execution of the dialogue box will produce a command that follows the syntax above. Since this command will automatically be saved in the session file for future reruns, one can use this description of the command to make modifications to the options without having to use the dialogue box interactively.

Example

```
dynaopts term 1. velocity 0. 0. 0. plti 1.  
          prti 1. nrest 20000 nrunr 1000 ;
```


IV. Frequently Asked Questions

1. How do I create an output deck for DYNA3D?

Go to the merge phase, issue a merge command, such as stp, issue the dyna3d command followed by the write command.

V. INDEX

a	26, 35, 36, 42	Body force
Acc	6, 9	material
Accc	10	
Accci	10	
Acceleration	9, 10, 48	Bold
prescribed	6	syntax
view	7	Boundary conditions
Acci	10	5, 8
Accs	10	nonreflecting
Accsi	10	6
alpha	28, 33, 40, 41, 43	si
angles	40	23
Angular velocity	48	view
Anisotropic	11, 12	Brick elements
aopt	32, 38, 39, 44	Bricks
Archive	7	Brode
Area	20	47, 48
Arri	6	Brode functions
Automatic contact	22	6
ax	33, 39-41, 44	Bsd
ay	33, 39-41, 44	5, 13, 19
az	33, 39-41, 44	Bt
b	5, 8, 26, 34, 36, 42	30, 31, 49
Base acceleration	48	C0
Base accelerations	6	30, 49
Bb	7	Carea
make	7	20
Bciz	30, 49	Clb
be	28	48
beam	5, 12, 13, 31	Co
beam cross section	21	7, 8
Beams	12	si
Belytschko-Schwer	20, 31	23
beta	28, 33, 34, 39-41, 44, 45	Comment
Bi	5, 8	7
Bind	5, 13	Concentrated loads
Block	5, 6	9, 10
Block boundary	7	Condition
Bm	6, 12	7
beam cross section	21	si
Bmcross	31	23
		Constitutive
		49
		Constraints
		8
		Contact Surface
		5-7, 21
		1D
		6
		faces
		8
		gap
		8
		graphics
		8
		initial penetration
		8
		merged nodes
		8
		mesh density
		8
		orientation
		8
		penalty
		48
		replication
		8
		sets
		8
		table
		8
		type
		7
		view
		7
		Contact surfaces

sii	23
Control cards	5, 56
Control phase	30, 46, 56
Convergence	49
Cpb	48
Cross section	5, 6, 11-13, 48
crushed	29
Csf	6
Ctb	48
Cylinder	5, 6
d	34, 41
Damper	13
Dampers	6
Damping	50
Data base	14
Database	14, 47-50
Default	21
Defpf	47
Delete	47
Detonation	6
Dtp	6
Dialogue box	
sid	22
spd	26
Displacement	10
Displacements	
prescribed	6
view	7
Dist	6
Distributed loads	9, 10
Dom	6
Drdb	50
Drflg	14
Dummy interface	22
Dyna3D	7, 26, 30, 47
Dynaeos	5, 26
Dynamats	30
beam cross section	21
Dynamic relaxation	49, 50
Dynaopts	5, 6, 14
analysis option	47
e	32, 33, 36, 38, 41-44
ea	32, 38-40, 42, 43
eb	32, 38-40, 42, 43
Edsdf	47
Elastoplastic	49
Element	
delete	47
sand	47
Element forces	47
Element history	6
Element type	49
Elfom	31
Elfor	30
EOS	
Energy deposition	26, 28
Gruneisen	26, 27
Ignition and growth	26, 28
JWL	26
Linear Polynomial	26, 27, 29
pore collapse	26, 29
Ratio of polynomials	26, 27
Sack	26, 27
tabulated	26, 29
tabulated w/ compaction	26, 29
Epb	6, 14
Equation of state	5
Experimental material	32
Extra nodes	6
Face sets	8
Facrx	49
Failure	6, 14
Fc	6, 9, 10
Fcc	9
Fcci	9
Fci	9
Fcs	9
Fcsi	9
Fd	6, 10
Fdc	10
Fdci	10
Fdi	10
Fds	10
Fdsi	10
Ffc	10
Fields, Richard	15

Flcd	6, 11	beam cross section	21
Fmom	10	Iff	14
Fn	6	Ihq	50
Fni	6	Iif	47
Follower force	10	Indentation	56
Follower forces	6	Inertia	20
view	7	Initial velocity	6
Force	47, 48	view	7
Force output	6	Integration	49
Format	7	Integration points	12, 13
Fr _b	6, 10	Integration rules	5
Fr _{bi}	10	Interface saved segment	47
Friction	22	Interface saved segments	6
Fset	8	Irr	20
F _v	6, 9, 10	Iss	6, 20
Fvc	9	Italicized	
Fvc _i	9	syntax	5
Fvi	9	Itr _x	49
Fvs	9	Itss	48
Fvsi	9	Itt	20
Fvv	9, 10	Jbm	6, 12
Fvv _c	9	beam cross section	21
Fvv _{ci}	9	Jbmi	12
Fvvi	9	beam cross section	21
Fvvs	9	Jd	6, 13
Fvvsi	9	Joints	6, 13
g	28, 33-37, 41	Jt	6, 13
gab	32, 39, 40, 42, 44	Kbm	6, 12
gama	34	beam cross section	21
Gravity	47	Kbmi	12
Gravity stress initialization	6	beam cross section	21
Grep	8	Lb	6
Gsii	8	Lcb ₁	48
Gvst	47	Lcb ₂	48
Hexahedral	11	Lcd	6, 11
History	48	Lcmax	49
Hiteb	47	Load curve	6
Hl	30, 31, 49	Load curves	11
Hour glass	50	Local constraints	8
Hughes-Lui	20, 31	Local system	8
Ibm	6, 12	Lrep	8
beam cross section	21	Lsii	8
Ibmi	12	Lsys	6

Make BB	7
Mass points	6
Mate	5, 11, 12
Material	
coordinate system	11, 12
Materials	5
body force	7
rotation	6
max	34-36
Mbb	7
Mdep	6
Membrane	6, 30, 49
Merge	6
dummy interface	22
nodes	7
Merged nodes	8, 13, 14
Mesh density	8
Mns	8
Mom	6, 9, 10
Moments	9, 10
Momentum	
deposition	6
Momi	9
Mpc	6, 13
Mt	5, 11, 12
Mti	5, 11, 12
Multiple constraints	6
Multiple point constraints	13
N	6, 12
shell orientation	5
Ndl	6, 9, 10
Ndli	9
Ngrav	48
NIST	15
Nodal constraints	5
view	7
Nodal forces	6
view	7
Nodal history	6
Node set	13
Node sets	8
Non-reflecting	8
Nonreflecting boundaries	6
view	7
Normal vector	12
Npb	6, 14
Npm	6, 13
Nr	6, 8
Nrest	48
Nri	6, 8
Nrunr	48
Nset	8, 13
Nseti	13
omega	27
Or	6, 11, 12
Orientation	8, 11, 12, 21
view	7
Orpt	8
Orthotropic	11, 12
Outout format	7
Parts	5
Plane	6, 8, 11
Plane strain	49
Plas	49
Plate	5
Plot data	48
Plti	14, 48
Pm	6, 13
Pnlt	48
Point mass	13
Point masses	6
Post processing	14
Pr	6, 9, 10
Pramp	6, 9, 10
Prescribed	6
Pressure	6, 9, 10
view	7
Pri	9
Print flag	47
Prism	11
Propt	30
Prtflg	50
Prti	14, 48
Pslv	8
Q1	50
Q2	50

Qh	50	Shth4	31
quad	30-32	Si	5, 6, 8, 23
Rayl	50	sid	22
Reaction force	47	Sid	6, 7, 21
Readmesh	7	with si and sii	22
Rebar		Sii	5, 6, 8, 23
sliding	22	Siid	5
Remarks	19	Sind	5, 12, 31
Replication	8	Single point constraintsts	6
Restart	49	view	7
Rfpf	47	Slide lines	21
Rho	32	Sliding boundary planes	6
Rigbm	6	view	7
Rigid bodies	6	Sliding interface	21
inertia	6	display	23
Rigid body	13	dummy interface	22
Rigid walls	6	si	23
Rotation	6, 9, 10	sii	23
material	6	Sliding Surface	5-7
Sand	47	1D	6
Sarea	20	faces	8
Scftrx	49	gap	8
Set ID	11	graphics	8
Sfb	6	initial penetration	8
Sfor	49	merged nodes	8
Shared constraints	6, 13	mesh density	8
view	7	orientation	8
shear	30-32	penalty	48
Shear area	20	replication	8
shell	5, 30	sets	8
Shell elements	49	table	8
Shell normal	49	type	7
Shell orientation	5	view	7
view	7	Sloc	31
Shell strain	49	Snrs	49
Shell thickness	5, 49	Solid to shell	6
Shell type	49	spall	35
Shells	11	Spd	6, 13, 25
Shloc	31	Spdp	6, 13
Shth	31	spd	26
Shth1	31	Spring	6, 13
Shth2	31	properties	25
Shth3	31	spd	26

Springs	6
Ssdm	14, 49
Ssf	12
Ssfi	12
Stabilization	50
stat	47
Statis reports	47
Sthi	19, 31
Sthi1	20, 31
Sthi2	20, 31
Stone wall	11
Stone walls	6
view	7
Stp	7
dummy interface	22
Strain	49
Stress	47
Stsm	47
Stss	49
Stup	49
Surface constraint	8
Sw	6, 11
Swi	6, 11
Syf	6
Symmetry	6, 8
view	7
Symmetry w/ failure	6
view	7
Syntax	56
T	7
dummy interface	22
Tb0	48
Te	6, 13
Tei	6, 13
temp	6, 13, 33
Temperature	6, 13
view	7
Teo	48
Tepro	6, 13
term	48
Termination	48
Tetrahedral	11
Th	5, 6, 12
Thermal effects	48
Thi	12
Thic	12
Thick shells	5, 12
Thickness	12
beam	19
Ticsf	48
Tie-breaking	6
Tied contact	21
Tied nodes	6, 14
view	7
Time history	6, 14, 48
Time interval	48
Time stamp	7
Time step	47, 49, 50
Time steps	
initial	48
restart file	48
saved restart file	48
Title	7
Tloc	32
tmm	7
Tolrx	49
Tp	7
dummy interface	22
Truss	5, 20, 31
Tshell	32
Tsmin	49
Tssf	49
Tsti	30
Tthi	20, 31
Tthi1	20, 31
Tthi2	20, 31
US Dept. Commerce	15
Vacc	10
Vaccc	10
Vacci	10
Vacci	10
Vaccs	10
Vaccsi	10
Variable acceleration	10
Variable thickness	12
Variable velocity	9, 10

Ve	6, 9
Vei	9
Velocities	9, 10
Velocity	6, 9, 48, 49
prescribed	6
view	7
virgin	29
Wedge	11
Write	7
Xb0	48
Xvel	48
Yase	30, 49
Yb0	48
Ylbd	47
Yvel	48
Zb0	48
Zvel	48