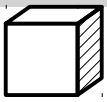
ELEMENTS

The following are a subset of available elements available in Abaqus. Sections refer to the Abaqus Analysis User's Manual.

CONTINUUM



CIN3D8
CIN3D18
CINAX4
CINPS4

C3D20RHT
- T/D/E/P
- hybrid
- R/I/M
- number of nodes
- 1D/2D/3D/PE/PS/PEG/AX/GAX
- C/DC/DCC/AC

§22.1

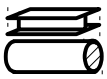
SHELL

S3
S4RSW
STRI3

S8R5W
- warping
- 5/T/S
- reduced integration
- number of nodes
- S/SC/STRI/DS/SAX/SAXA

§23.6

BEAM



B31OSH

B31OSH
- hybrid
- open section
- order, 1/2/3/4
- dimension, 2/3
- B/PIPE

define cross-sectional profile and relative beam orientation §23.3

RIGID/ANALYTICAL

R2D2
RB2D2
R3D4

RB3D2
- number of nodes
- 2D3D/AX
- beam
- R

§24.3

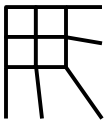
MEMBRANE/SURFACE

M3D4R
MGAX2

M3D4R
- reduced
- number of nodes
- 3D/G+AX/CL

§, §26.7

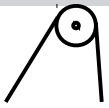
INFINITE/SEMI-INFINITE



_IN_
- infinite

§22.2

CONNECTOR



CONN2D2

CONN3D2
- number of nodes
- 2D/3D
- connector

§25

TRUSS

T2D2
T3D3H
T2D2T
T3D3T

T3D3H
- hybrid
- number of nodes
- displacement
- dimension, 2/3
- C/DC/DCC/AC

§23.2

POINT

MASS
HEATCAP
ROTARY INERTIA

§24.1-2,4

ACOUSTIC/HYDRODYNAMIC

ASI1
AC1D3
F3D3
FLINK

ASI3D8
- number of nodes
- 1/2D/3D/AX
- ASI; see also AC (continuum)

§26.14

F3D3
- number of nodes
- 2D/3D/AX

§26.8

GAP/CONTACT

GAPUNI
DGAP

§26.5, §31

SPECIAL-PURPOSE

SPRING
DASHPOT
JOINT

include springs, dashpots, joints, gaskets, drag chains, pipe–soil, coupling elements §26

Topologically, `CPE4=CAX4R=S4R=DC2D4`, etc.; Abaqus/CAE does not check DOFs.

DEGREES OF FREEDOM

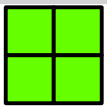
1 x-displacement
2 y-displacement
3 z-displacement
4 rotation about x-axis, radians
5 rotation about y-axis, radians
6 rotation about z-axis, radians
7 warping amplitude (for open-section beam elements)
8 pore pressure, hydrostatic fluid pressure, or acoustic pressure
9 electric potential
10 connector material flow, length
11 temperature (or normalized conc. in mass diffusion analysis)
12 second temp. (shells or beams)

MESHING

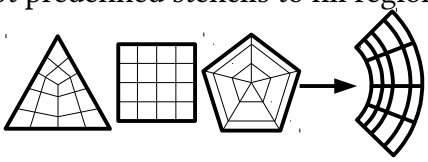
METHOD

1. Mesh *independent* or *dependent* part instance
2. Assign mesh controls · seeding, element type, meshing technique
3. Generate
4. Refine · with goal of fast accurate convergence
5. Verify · using verification tools
6. Optimize · based on analysis results

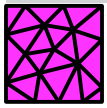
STRUCTURED



Adapt predefined stencils to fill region.



FREE



Do not use a preestablished mesh pattern; mesh is determined by region topology and other elements. (Quad/Tri/Tet only)

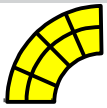
ADVANCING-FRONT

Generate elements on boundary, proceed inwards on same basis

MEDIAL AXIS

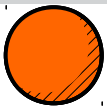
Use internal partitions to seed simpler regions

SWEPT



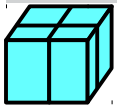
Generate mesh on a face (edge) and then sweep along a sweep path through the volume (area).

UNMESHABLE



Use partitions and/or the bottom-up technique to render an unmeshable part manageable.

BOTTOM-UP



Build within region like building blocks (not constrained to fill specific geometry).

IMPORTING & CONVERTING SOLID MODELS

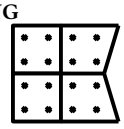
Elysium translates most CAD formats to Abaqus-compatible formats; prefer `CAE`, `IGS`, `SAT`, `ENF`.

MESH QUALITY

SELECTION CRITERION	QUADRILATERAL	TRIANGLE	HEXAHEDRON	TETRAHEDRON	WEDGE
SHAPE FACTOR	N/A	0.01	N/A	0.0001	N/A
SMALLER FACE CORNER ANGLE	10	5	10	5	10
LARGER FACE CORNER ANGLE	160	170	160	170	160
ASPECT RATIO	10	10	10	10	10

ISSUES

HOURGLASSING



CAUSE

Reduced integration can lead to hourglassing, in which undersampling can lead to uncontrolled degrees of freedom which oscillate freely.

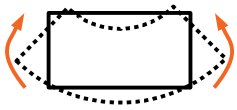
DIAGNOSIS

This is most easily detectable visually, when elements alternate in structure.

CORRECTION

Use hourglass control or avoid the use of reduced-integration elements. Refine the mesh in regions of large plastic strain.

SHEAR LOCKING



CAUSE

Manifests in first-order fully-integrated elements as a nonphysical numerical stiffness; this creates shear strains that make elements too stiff in bending.

DIAGNOSIS

Excessive shear strain which mesh refinement does not remove.

CORRECTION

Use enhanced strain or extra shape functions; avoid long thin bending elements.

VOLUMETRIC LOCKING

CAUSE

Occurs in fully-integrated elements with near-incompressible material behavior; spurious pressure stresses develop at integration points, leading to overstiffness.

DIAGNOSIS

Pressure stress at integration points shows checkerboard pattern.

CORRECTION

Refine the mesh in regions of large plastic strain.