Abaqus theory guide 6.14 pdf

The answer to the OverviewA project is one scalable value, such as the size of your structure; Calculated by the optimization module by reading the results and model data from the output database file. can be transferred from objective functions and limitations (for example, you can create an objective function that tries to minimize the movement to the node or restriction, which causes to reduce the weight of the structure by at least 50%); certain analysis procedures (for example, when choosing a response to a design that tries to maximize the lowest eigenfrencies, you need to perform an analysis of the extraction of eigenvalue). although some restrictions apply. For example, a volume design response can only use the amount of volume in the design area. The project response, calculated by von Misanza's stress, should take the maximum value of stress in the model region. (None of the operators matter when the optimization module calculates the dynamic design response of the frequency.) Operators of responses to the following projects are provided by the optimization module: Minimum or maximum: minimum or maximum value in the chosen region. The optimization module allows only the maximum operator for stress, contact stress, and voltage design responses. Amount: The sum of all values in the selected area. The optimization module allows only the operator of the amount for volume, weight, moment of inertia and reaction of gravitational design. Project responses to optimize

topology based on conditions The Optimization Module provides a load of energy and a volume of project responses for state-based topology optimization. Strain energyMeservation structure is a measure of its overall flexibility or rigidity and is defined as the amount of voltage energy of all elements, for linear models where the vector shifts and is a global rigidity matrix. Compliance is mutual rigidity, and minimizing compliance is equivalent to maximizing global rigidity. If the load casing is caused by force or pressure, you should choose to minimize energy voltage to maximize global stiffness. However, if the load casing is controlled by a heat field, the voltage energy decreases when the optimization changes the structure to make it softer. As a result, you should always choose to maximize energy voltage because trying to minimize energy voltage can lead to a rigid structure. In addition, you should always choose to maximize energy voltage if prescribed biases apply to your model. Topological optimization takes into account the total voltage energy for all elements; so if you choose voltage as an objective function, you have to apply the goal to the Model. You can't use voltage energy as a limitation in optimization. Abaqus/CAE Use: Optimization Module: Task-based task topology, Design ResponseCreate: One-term, Variable Energy Strain VolumeThe volume is defined as the amount of elements in the design area where the volume of the item is located. While optimizing the topology, the elements scale with the current relative density defined in your Abaqus model. For most optimization issues, you need to apply a volume limit. For example, if you're trying to minimize energy voltage (maximise rigidity) and don't apply a volume limit, the optimization module simply fills the entire design area with the material. Abaqus/CAE Use: Optimization Module: Taskcondition-based Topology Challenge, Design ResponseCreate: One-Term, Variable: Volume Design Responses for General and Size Topology Optimization Module Provides Center of Gravity, Displacement, Rotation, Eigenfrequency, Energy Rigidity, Moment of Inertia, Internal and Reactionary Forces and Moments, Energy Voltage, Volume Center of GravityYYy can use the center of gravity of the chosen area as a design response in optimization. You can choose the center of gravity in three main directions: When the optimization module calculates the center of gravity, the elements scale with the current relative density defined in your Abaqus model. For example, you can limit the center of gravity in the direction of Y so that it stays within the minimum and maximum range during optimization. A design reaction can consider the center of gravity of the entire model or the scope of the model. When selecting a local coordinate system, the optimization module uses both the direction of the axes and the position of the source to recalculate the center of gravity. The optimization module uses a global coordinate system if you don't choose a local coordinate system. When the optimization module calculates the center of gravity, it considers the shell and membrane areas as three-dimensional areas, applying the thickness of the area. The optimization module calculates the center of gravity using only the types of elements supported by topology optimization. As a result, the center of gravity calculated by the optimization module may not be the same as the center of gravity calculated by Abaqus/Standard or Abaqus/Explicit; for example, if the model contains wire areas. Abaqus/ CAE Use: Optimization Module: Taskgeneral Topology or Task Size, Design ResponseCreate: One-Term, Variable: Gravity and Rotation CenterIn most optimization problems you'll use and/or rotations to determine your objective function or limitations. For example, maximizing the movement of the top can be either a goal or a limitation of optimization. Optimization performance improves if you apply bias and rotation only to verticals or to small regions. In addition, performance improves when assigning regions that are used to identify biases or reactions as frozen regions (the optimization module will not remove items from frozen regions during optimization). Table 13.2.1-1 lists available variables of movement and rotation. Table 13.2.1-1 Variable offsets and rotations for general and dimensional optimization of topology. MovingRotationi-directionAbsoluteAbsolute in i-direction Abaqus/CAE Use: Optimization module: Topology task or task size, Design ResponseCreate: One-term, Variable: Moving Energy Rigidity measureThe energy rigidity is a measure that makes no physical sense, but can be used as an objective function or limitation in overall topology or optimization of the size to optimize the rigidity of the structure that is exposed as an external To optimize the rigidity of the structure only with external load, the energy voltage should be minimized: where the external load and as a result of the deviation of the loaded nodes. If there is only an external load, the energy stiffness measure is equal to the total voltage energy, also called conformity. In contrast, if the load casing is conditioned by prescribed movements, elastic energy or compliance is reduced only if the structure becomes softer. To optimize a structure with only prescribed movements, voltage energy should be maximum: where shift in nodes is prescribed and is the resulting reaction force on the displaced nodes. If only prescribed movements are present, the energy stiffness measure is equal to negative from the total voltage energy. Voltage energy with both external loads and prescribed movements is given, as the energy rigidity measure is only used for optimization (it has no physical value) and is given because you cannot use the energy rigidity measure as a construct reaction in a model that experiences heat load or mass-dependent load, such as gravity. The energy rigidity design reaction should be applied to the entire model. When energy rigidity is used as an objective function, you need to choose a target that tries to minimize the amount of weighted differences between the design reaction and the reference value, regardless of whether the external load and/or prescribed movements apply to the structure. Abaqus/ CAE Use: Optimization module: Topology or Size Task, Design ResponseCreate: Single-, Variable: Energy Stiffness Measure Modal Modal analysisModal eigenvalues are the simplest dynamic responses in structural analysis. Typical use of data from the eigenfrequency analysis during topology optimization include: maximize the lowest eigenfrequencies, maximize selected eigenfrequency, limit eigenfrequency to be higher or lower than this value, maximize or minimize eigenequfrency in a certain mode, and perform a range optimization that forces modes from a certain frequency. The optimization module supports two approaches to optimizing eigenfrequencies: unified eigenfrequencies from modal analysis and Kreisselmaier-Steinhauser formulation. The wording of Kreisselmaier-Steinhauser is the more effective of the two approaches and should be used as far as possible. The only advantage of optimizing single eigenfrequencies is that you can use the amount of eigenfrequencies as a limitation in overall topology or optimizing sizes that you can't do with the wording Kreisselmaier-Steinhauser. When you try to maximize the lowest eigenfrequency, it is recommended to consider not only the first eigenfrequency, but also at least the following two highest natural frequencies. During optimization, different natural frequencies are weighed at a distance from the lowest natural frequency - the closer the natural frequency approaches the first natural frequency during optimization, the more it is weighted. You should use the wording Kreisselmaier-Steinhauser eigenvalue if you are trying to maximize the lowest eigenfrequency or in particular if you are trying to maximize more than one of the lowest eigenfrequencies. You don't need to use tracking mode if you use the Kreisselmaier-Steinhauser formulation to maximize the lowest eigenfrequency, but you should use tracking mode for higher modes because modes can switch. For example, while optimizing the model, the frequency of the first mode is maximized, and the second eigenmode may become the lowest-frequency mode. Abaqus/ CAE Use: Optimization module: Taskgeneral topology or task size, Design ResponseCreate: One-term, Variable: Eigenfrequency from modal analysis or Eigenfrequency calculated with Kreisselmaier-Steinhauser Formula Moment of InertiaYYOu can use the moment of inertia design response in optimization to minimize rotation inertia on the chosen axis. The amount of inertia of the entire model or model area can be used as an objective function or limitation in the overall topology or optimization of sizes. You can choose the moment of inertia in three main directions or three main planes: If you choose a local coordinate system, the optimization module uses the direction of the axes to recalculate Gravity. The optimization module applies a global coordinate system if you Don't select a local coordinate system. When the optimization module calculates the moment of inertia, it considers the shell and membrane areas as three-dimensional areas, applying the thickness of the area. The optimization module calculates the moment of inertia using only the types of elements supported by topology optimization. As a result, the moment of inertia calculated by the optimization module may not be the same as the moment of inertia calculated by Abaqus/Standard or Abaqus/Explicit; for example, if your model contains beam or farm elements (wires in Abaqus/CAE). The moment of inertia against any two orthogonal axis is zero if you have chosen one of the axes to be the axis of symmetry. Abaqus/CAE Use: Module Optimization: Taskgeneral Topology or Size Task, Design ResponseCreate: One-Term, Variable: Moment of Inertia Internal Forces and MomentsYit can use the present internal forces and moments of the entire model or model area as an objective function or limitation in overall topology or size optimization. Table 13.2.1-2 lists available internal force variables and moment. Table 13.2.1-2 Knot inner strength and instant variables for general and dimensional topology optimization for e elements attached to i nodes. Your structure should be rigid in the direction of the force used in optimization; otherwise, the internal force will be zero in that direction. Abaqus/ CAE Use: Optimization Module: Taskgeneral Topology or Task Size, Design ResponseCreate: One-Term, Variable: Internal Forces or Inner Force Reaction Force and MomentsNodal Reaction Force and Moments can be used as a response to design only in general and the size of topology optimization. As with moving, optimization performance improves if you apply reaction forces or moments only to vertical or small regions and assign those regions to frozen regions (the optimization module will not remove items during optimization). Table 13.2.1-3 lists available variable reaction forces in the nod and moments. Table 13.2.1-3 Reaction Force and Instant Variables for the general and dimensional topology of e-optimization elements attached to i nodes. Your structure should be rigid in the direction of the force used in optimization; otherwise, the reaction force would be zero in that direction. Abaqus/CAE Use: Optimization Module: Topology of Tasks and Size Challenge, One-term, Variable: Reaction Force or Reaction Moment Strain EnergyThe conformity of the structure is a measure of its overall rigidity and is defined as the amount of voltage energy of all elements, for linear models where the vector of displacement and global rigidity of the matrix. Compliance is mutual rigidity, and minimizing compliance is equivalent to maximizing global rigidity. If the load casing is controlled by a heat field, the energy voltage decreases when the structure becomes softer. As a result, trying to minimize energy voltage can lead to a rigid structure. In addition, you should always choose to maximize energy voltage if prescribed biases apply to your model. Topological optimization takes into account the total voltage energy for all elements; so if you choose voltage energy as an objective function, you should apply the target to the entire model. Abaqus/CAE Use: Optimization Module: Taskgeneral Topology or Task Size, Design ResponseCreate: One-Term, Variable: Energy Strain Scaled by Centroid von Misus stressYou can use the scalable element of centroidal von Misanza stress throughout the model or model area as an objective function or as a limitation in overall topology optimization. Regions with stressful singularities caused by external loads or border conditions should be avoided. The scalable element of centroidal von Mizancia stress is defined as where the element of centroidal von Misanza stress is a reference stress, and is a factor for interpolating the stresses of elements that have a decrease in current relative density due to the optimization of topology. Weight coefficient and interpolation are necessary for convergence during optimization. Von Misanza's stress is calculated on the element's centroid to avoid stressful singularity that may be present in the original model or may appear in an optimized structure before it is smoothed out. You can't compare the scalable element of centroidal von Misanza stress with von Miz stress calculated by Abaqus. These two measures are equal only when the element is solid and has a relative density of 1.0.You can provide reference stress when creating an objective function, or the optimization module can calculate reference stress during the initial optimization iteration. If you provide background stress, the value should not be too low or numerical singularity will result. Reference stress is given as you can identify several cases of load for the scalable element of centroidal von Misanza stress measures. Static linear analysis is supported. Static non-linear analysis only supports non-linear contact. Nonlinear materials and geometric nonlinear phenomena, such as large deformations, are not supported. Use: Optimization module: Taskgeneral topology, Design ResponseCreate: One-term, Variable: Variable: Volume volume is defined as the amount of all items in the design area where the item is located. For most optimization issues, you need to apply a volume limit. For example, if you're trying to minimize energy voltage (maximise stiffness) and don't apply volume limit, the optimization module simply fills the design area with the material. Abaqus/ CAE Use: Optimization module: Taskgeneral topology or task size, Design ResponseCreate: One-term, Variable: Weight Weight Weight Is Defined as the Weight Of All Elements in Design Where the Weight of an Element is located. The optimization module scales the elements using current relative density. For most optimization issues, you need to apply a volume or weight limit. Using weight instead of volume allows you to limit the optimized model to a given physical weight. The optimization module only uses supported types of items when calculating weight. Abaqus/CAE Use: Optimization Module: Taskgeneral Topology or Size Task, Design ResponseCreate: One-Term, Variable: Weight Design Responses to Optimize Form Optimization Module provides eigenfrequency, stress, contact stress, voltage, nodal energy density strain, and volume of design responses to optimize form. You can only use the reaction to the volume design to determine the limitation; all other design responses are used to determine objective functions. Eigenfrequency from the wording Kreisselmaier-SteinhauserOu should use the wording Kreisselmaier-Steinhauser eigenvalues as an objective function in optimizing the form if you are trying to maximize the first eigenfrequency or in particular if you are trying to maximize more than one of the first eigenfrequencies. You don't need to use tracking mode if you use Kreisselmaier-Steinhauser wording eigenvalues. Abaqus/CAE Use: Optimization Module: Task Challenge, Design ResponseCreate: Single-Term, Variable: Eigenfrequency calculated with the Kreisselmaier-Steinhauser formula Stress and Contact StressCem is the most commonly used objective form optimization feature. All voltage values calculated by the optimization module, whether node or from Gauss points or elements, are interpolated into nodes. For example, you can try to optimize the model with an objective feature that tries to minimize von Misans' maximum stress in a region with a concentration of stress or tries to minimize contact pressure in a region with contact. The optimization module only takes into account the maximum value of equivalent stress in the region. The optimization module issues warnings for nodes that do not have the appropriate voltage values. For example, if you choose response to contact stress, the optimization module issues warnings about items that are not in contact. If the Abaqus model contains multiple downloads, the design response is evaluated by summing up the voltage values in each load case. You can choose from the following equivalent stresses: von MisesMaximum basic and absolute maximum principleMinimum main and absolute minimum principalSecond principalBeltramiDrucker PragerGalileiKuhnMariotteSandelSauterTrescaOu you can choose one of the following equivalent contact stress:Contact pressure Total haircut contact stressSher contact stress in 1-directionHer contact stress in 2-directionTotal contact stressY can create a design response that uses stress or contact stress only in the form of optimization, and it can only be used as an objective function. Abaqus/CAE Use: Optimization Module: Task Challenge, Design ResponseCreate: Single-Term, Variable: Stress or Contact Stress StrainIf Your Model Undergoes Large Strain, a measure of stress is not always a good indicator of the model's response. For example, the structure undergoing plastic deformation will, for the perfect plastic material, experience great constant stress over the plastic area. In these conditions, the deformation measure is a more reliable indicator of the model's reaction. You can choose from the following equivalent strains: ElasticPlasticTotal (the amount of elastic and plastic) you can create a design response that uses voltage only in optimizing the form, and it can only be used as an objective function. Abaqus/CAE Use: Optimization Module: Task Tasks, Design ResponseCreate: Single-Term, Variable: Strain of Nodal Strain energy energy density nodal, is a local point-wise strain of energy that can provide a better view of failure than stress in nonlineous materials. Abaqus/CAE Use: Optimization Module: Task Challenge, Design ResponseCreate: Single-Term, Variable: Volume Volume Energy Density Voltage is the only limitation allowed to optimize the form. Volume is defined as the amount of all items in the design area where the item is located. To solve most optimization issues, you need to apply a volume limit to the model area. For example, if you're trying to minimize energy voltage (maximise stiffness) and don't apply a volume limit, Abaqus simply fills the design area with the material. Abaqus/CAE Use: Optimization Module: Task Challenge, Design ResponseCreate: One-Term, Variable: Volume Working on Project ResponsesYSYS, You can determine a design response that is a combination of single values generated by multiple design responses; for example, you can add values or find a maximum of several values. You can also A response to a design that is the result of an operation on a different design reaction. for example, the difference between the value of the answer to design design different knots. For example, you can create two design reactions that correspond to a one-way shift between the two selected vertices. You can also create a design response that is the difference between moving 1-direction to the two selected vertices. You can then determine the limitation that causes the structure to react to be close to zero. In fact, the restriction forces the two selected vertices to move together in a one-way direction. Abaqus/CAE Use: Optimization Module: Design ResponseCreate: Combined Term abaqus theory guide 6.14 pdf