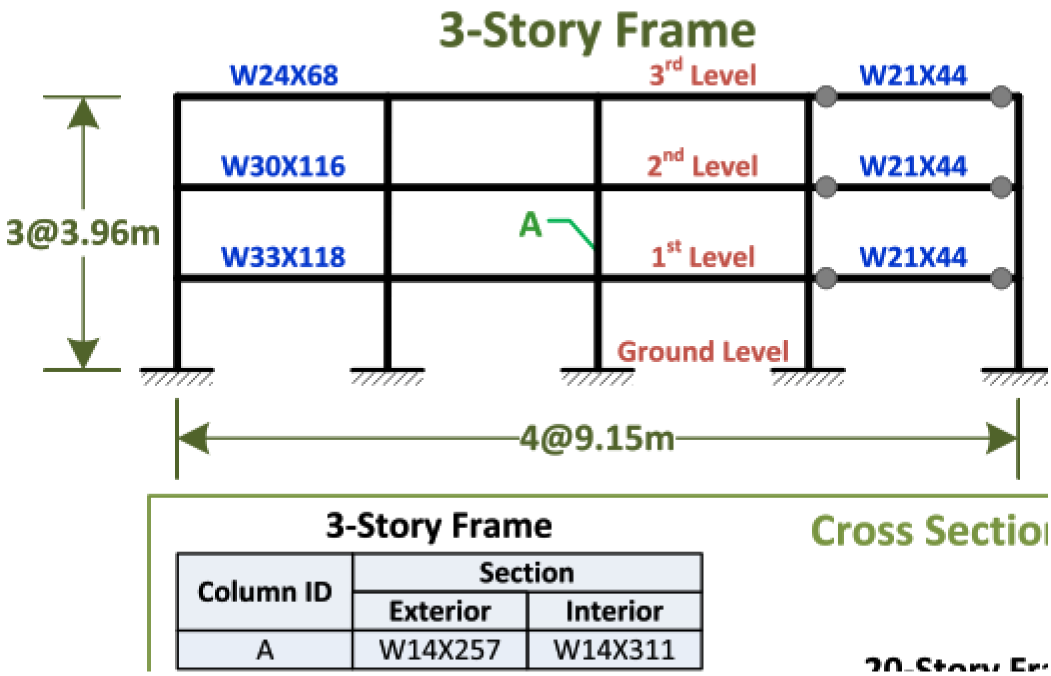
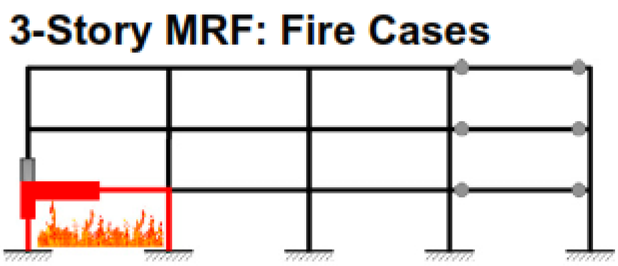
بسمه تعالی

# تحلیل قاب خمشی فولادی در آتش

در این پروژه یک قاب سه طبقه با چهار دهانه مدل سازی شده سپس دوره تناوب های این قاب محاسبه می شود. در ادامه در یک مدل جدید بارگذاری های ثقلی اعمال شده و سپس در یک گام تحلیل ویسکوز شار حرارتی به صورت یک نمودار که شامل افزایش دما و کاهش دما می باشد به تیر و ستون های یک محفظه اعمال می شود. در گام تحلیل ویسکوز پارامترهای خزش باید تعریف شود و خزش در نظر گرفته شود.




نکته‌ای که در این مدل سازی وجود دارد این است که در دهانه ای که آتش سوزی قرار است مدل شود باید المان های تیر و ستون ها همچنین اتصالات از نوع solid باشد اما سایر اعضای قاب باید دارای المان Beam باشند.

## 3. Multi-resolution finite element model of SAC frames

The commercial finite element software, ABAQUS v6.10 [50], is employed to create the numerical model of the MRFs. To evaluate the global response of the MRFs and local behavior of the selected RBS connections, a multi-resolution numerical model of the systems is created as shown in Fig. 3. The selected RBS connections are modeled using solid continuum elements while the remainder of the frames is modeled using 1-D line elements. The 3-D connections are extended to the mid-span and mid-height of the corresponding beam and columns, respectively. This allows for the evaluation of the influence of the surrounding structure on the behavior of the RBS connections. In locations where 3-D continuum model of the connection is not used, piece-wise reduced section approximation of the connection is utilized as shown in Fig. 3.

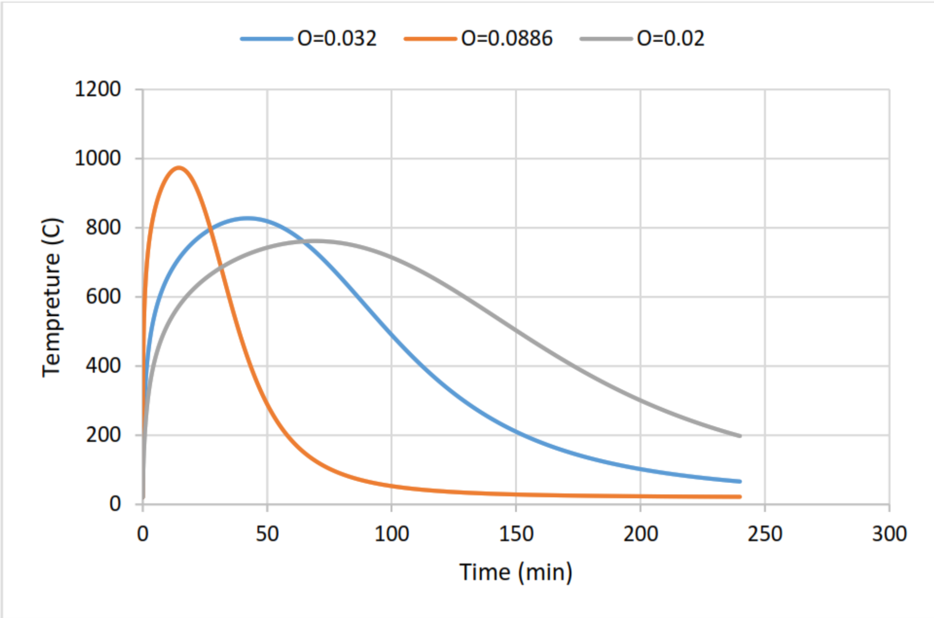
An uncoupled thermal-mechanical analysis is conducted to evaluate the system response under the applied fire cases. First, transient nodal temperatures are determined through running a thermal analysis. Afterwards, these nodal temperatures are imported to the mechanical analysis to obtain the corresponding stress and strain fields. ABAQUS/ Standard is employed to run the mechanical analysis using implicit static module [50]. It is noted that prior to importing the transient nodal temperatures, gravity loads were applied to the frames. In the thermal analysis, 8-node linear continuum heat transfer elements (DC3D8) and 2-node heat transfer link elements (DC1D2) are applied to the 3-D solid and 1-D line elements, respectively [50]. In the mechanical analysis, 8-node linear continuum reduced integration elements (C3D8R) are used for the 3-D solid elements, and 2-node linear beam elements (B31) are employed for the 1-D line elements [50]. Furthermore, multi-point constraint (MPC) connector, type TIE, is used to connect the thermal degree of freedom (NT11) in the line elements to the solid elements in the thermal model. In the mechanical model, MPC constraint, type BEAM, is implemented to tie all transitional and rotational degrees of freedom in the line elements to the solid elements. A mesh convergence study is performed on the 3-D RBS connection to ensure accuracy of the results, particularly in the critical zones, i.e. CJP welds, weld access holes, and RBS region. The mesh size is gradually increased with distance from the critical zones using various mesh techniques to ensure computational efficiency of the models. The length of line elements is smaller than 150 mm in all frames. In the 3-D RBS connections, the mesh size in the critical zones is smaller than 10 mm and increases to a maximum of 120 mm at the ends of the beam and column (Fig. 3).

The stiffness of the panel zone is modeled using the scissors model [51]. The panel zone – shown in Fig. 3 – is modeled by two rigid links that are hinged together at the mid-point, and connected to the remainder of the frame by the beam connectors. A rotational spring with temperature-dependent stiffness that is proportional to the beam and column sizes connects the two rigid links together. The P-Δ effects on the MRFs, associated with vertical loads on the interior gravity frames per tributary area, are taken into consideration using leaning columns as highlighted in Fig. 3. The leaning columns are axially stiff, elastic, and pinned at the basement and story levels. They are also connected to the main MRF at each story level using MPC, type LINK, from ABAQUS constraints library [50] as show in Fig. 3. The gravity loads are applied as concentrated loads at the story levels on the leaning columns.

The 3-D RBS connection requires to be braced to prevent out of plane rotation, particularly under inelastic behavior, in accordance with the *AISC 341-10 Seismic Provision* [52]. This includes restraining the bottom flange in the negative moment region, using bracing elements, and the top flange in the positive moment region, which is naturally provided by the concrete slab. This was realized through utilizing appropriate out-of-plane boundary conditions to restraint lateral-torsional buckling according to the *AISC 341-10 Specification* [52].

Temperature-dependent stress-strain behavior of the structural steel, A572-Gr50 [53], is adopted per Eurocode 3 [3]. Furthermore, ductile damage initiation in the steel material is captured through using the Johnson and Cook damage model [54]. A linear damage evolution is also employed to capture full degradation in the material. Further details regarding the adopted temperature-dependent material properties, damage initiation and evolution have been discussed in Memari and Mahmoud [11].

منحنی آتش مورد استفاده در این پژوهش نمودار خاکستری زیر است




شکل ۲۳ مدل آتش‌سوزی‌های مختلف

چند سوال در این پروژه وجود دارد که حتماً به آنها باید پاسخ داده شود:

۱. مدل آسیبی که دراین مقاله استفاده شده است چگونه است؟

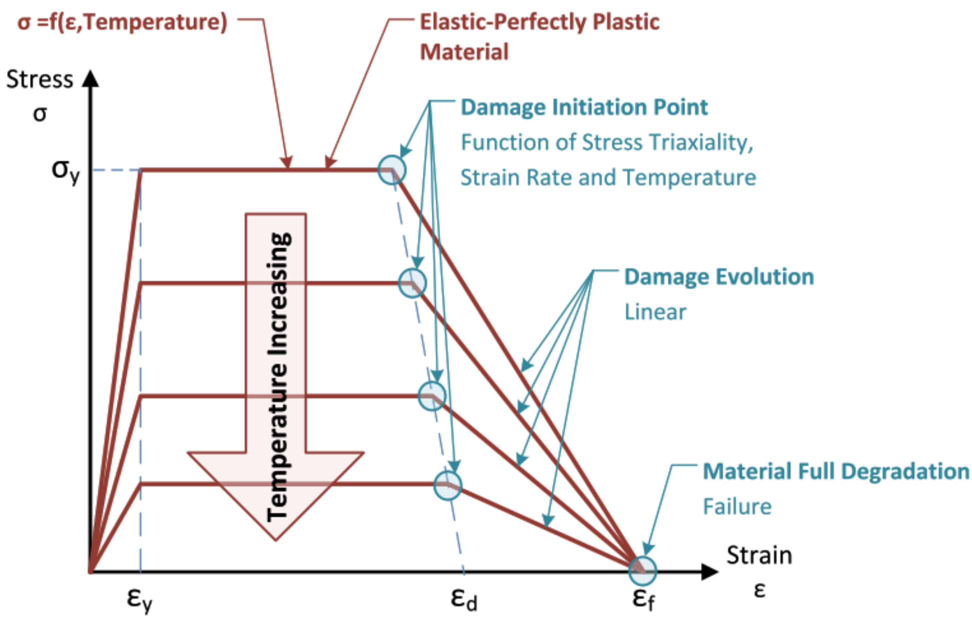



**Fig. 5.** Stress–strain relationship and damage modeling.

### *3.2. Material damage modeling*

An accurate prediction of the plastic deformation and failure of the steel elements essentially requires proper capturing of material damage. Damage initiation is modeled using the Johnson–Cook damage model [47]. The ductile damage initiation criterion defines the equivalent plastic strain ($\varepsilon_d$) as a function of stress triaxiality ($\sigma^*$), strain rate ($\varepsilon^*$), and homologous temperature ($T^*$) as shown in Eq. (1), where $\sigma^* = \sigma_m/\sigma$, in which $\sigma_m$ is the average of the three normal stresses and $\sigma$ is the von Mises equivalent stress.

$$\epsilon_d = (D_1 + D_2 exp(D_3\sigma^*))(1 + D_4 ln\,\epsilon^*)(1 + D_5 T^*) \quad (1)$$

The five constants $D_1$–$D_5$ are defined in the literature [47]. Furthermore, a linear evolution of the damage variable with effective plastic displacement is considered, which allows the effective plastic displacement to be specified at the point of full degradation. It is worth noting that the linear damage evolution, defined in this study, represents a linear stress–strain softening response because of the elastic–perfectly plastic material definition [44]. The elastic–perfectly plastic temperature-dependent material definition and the damage model are shown in Fig. 5.

۲. اتصال مورد نظر که با ۲عضو صلب که به صورت قیچی به هم وصل شده اند و یک فنر پیچشی آنها را به هم متصل می کند چگونه مدل سازی می شود؟

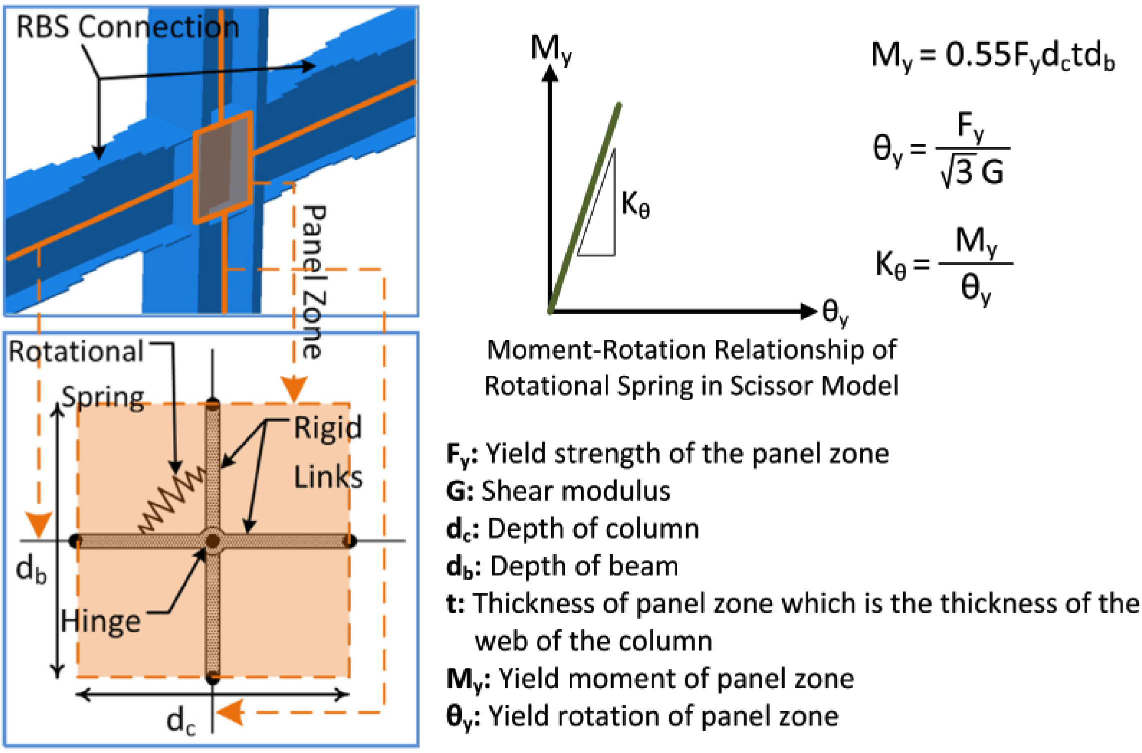



**Fig. 3.** Details of implemented scissor model to represent the panel zone.

۳. درین این پروژه چند نوع اتصال وجود دارد که توضیح نحوه مدلسازی آنها لازم است. یکی اتصال المانهای سالید به فرمان های خطی. بعد از آن اتصال المان صلب پنل زون به المان های خطی تیر و ستون که بعضی از آنها اتصال صلب هستند و بعضی از اتصال مفصلی

در مقاله از این اتصالات با نام multi-point constraint (MPC) connector, type TIE و MPC, type LINK, from ABAQUS constraints library عنوان شدند.

اگر لازم است تا متن انگلیسی را ترجمه کنم

من خودم مشخصات لازم برای مدل فولاد را تعریف کرده ام و در اختیار شما قرار میدهم